



**INVENTOR<sup>®</sup>**

Shortcut Key Guide



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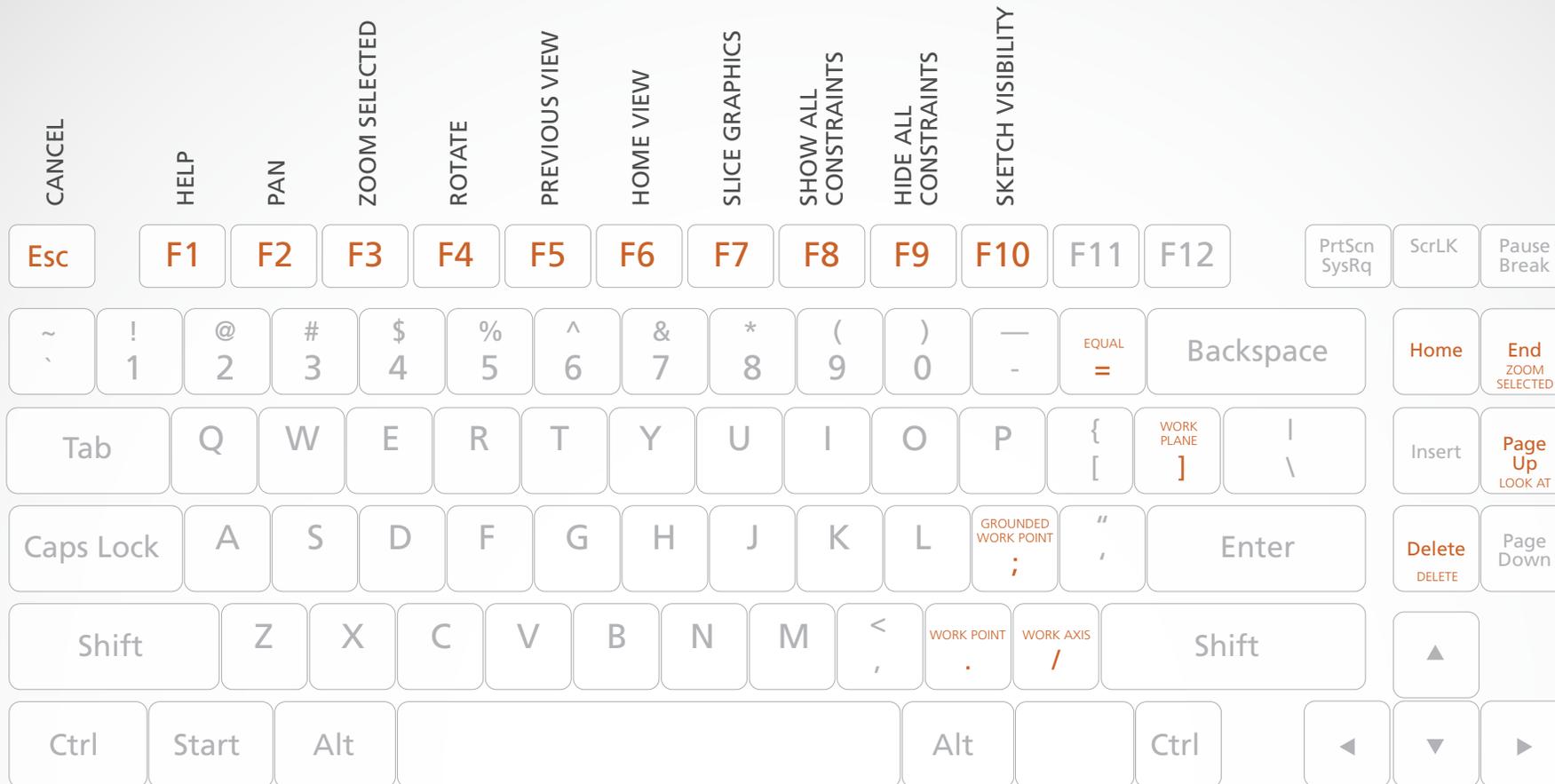
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**NOTE:** To enable the use of multi-character command aliases in Inventor, make sure the "Use default multi-character Command Aliases" option is checked. To access this, select **Tools** tab ▶ **Options** panel ▶ **Customize**, and click the **Keyboard** tab.



- = **EQUAL** / Constrains curves to equal radius or length.
- ; **GROUNDING WORK POINT** / Creates a grounded work point and activates the 3D Move/Rotate tool.
- / **WORK AXIS** / Creates a new work axis.
- ] **WORK PLANE** / Creates a new work plane.
- . **WORK POINT** / Creates a new work point.

- DELETE** **DELETE** / Removes the selected item from the model or drawing.
- END** **ZOOM SELECTED** / Zooms in on a specifically selected item.
- ESC** **CANCEL** / Closes the active tool.
- HOME** **ZOOM ALL** / Zooms to the extents of all objects in the graphics window.
- PAGE UP** **LOOK AT** / Looks at a selected entity.

## Assembly

### ALT-DRAG MOUSE

**MATE CONSTRAINT** / Applies a mate constraint.

### CTRL-H

**REPLACE COMPONENT** / Replaces one assembly component with another component.

### SHIFT-TAB

**PROMOTE** / Removes a part from a subassembly and makes it an individual part in the parent assembly.

## Drawing Manager

### CTRL-SHIFT-N

**NEW SHEET** / Inserts a new drawing sheet.

### CTRL-SHIFT-T

**LEADER TEXT** / Adds notes with leader lines to a drawing.

## General

### CTRL-A

**SELECT ALL** / Selects everything on the screen.

### CTRL-C

**COPY** / Copies selected items.

### CTRL-F

**FIND** / Presents the Find Part Sketches dialog box in part files; opens the Find Assembly Components dialog box in assembly files.

### CTRL-N

**NEW** / Displays the New File dialog box.

### CTRL-O

**OPEN** / Displays the Open dialog box, to open an existing file.

### CTRL-P

**PRINT** / Displays the Print dialog box.

### CTRL-S

**SAVE** / Displays the Save As dialog box.

### CTRL-V

**PASTE** / Pastes a cut or copied item from one location to another.

### CTRL-X

**CUT** / Cuts selected items.

### CTRL-Y

**REDO** / Redoes previously undone commands.

### CTRL-Z

**UNDO** / Undoes the effects of commands.

### DELETE

**DELETE** / Removes the selected item from the model or drawing.

### QUIT

**EXIT** / Quits the application; prompts to save documents.

### SHIFT-RIGHT MOUSE CLICK

**SELECT** / Activates the select command menu.

## Part

### CTRL-SHIFT-K

**CHAMFER** / Activates the Chamfer tool.

### CTRL-SHIFT-L

**LOFT** / Activates the Loft tool.

### CTRL-SHIFT-M

**MIRROR** / Activates the Mirror tool.

## Placed Features

### CTRL-SHIFT-O

**CIRCULAR PATTERN** / Opens the Circular Pattern dialog box.

### CTRL-SHIFT-R

**RECTANGULAR PATTERN** / Opens the Rectangular Pattern dialog box.

### CTRL-SHIFT-S

**SWEEP** / Activates the Sweep tool.

## Sketch

=	<b>EQUAL</b> / Constrains curves to equal radius or length.
F7	<b>SLICE GRAPHICS</b> / Slices away temporarily the portion of the model that obscures the plane.
F8	<b>SHOW ALL CONSTRAINTS</b> / Displays all constraints.
F9	<b>HIDE ALL CONSTRAINTS</b> / Hides All constraints.

## Tools

ALT-F8	<b>MACROS</b> / Opens the Macros dialog box.
ALT-F11	<b>VISUAL BASIC EDITOR</b> / Launches the Microsoft Visual Basic Editor.
ESC	<b>CANCEL</b> / Closes the active tool.
F1	<b>HELP</b> / Opens the Inventor help files.

## View

ALT-	<b>USER WORK POINTS VISIBILITY</b> / Makes work points visible.
ALT-]	<b>USER WORK PLANE VISIBILITY</b> / Makes work planes visible.
ALT-/	<b>USER WORK AXES VISIBILITY</b> / Makes work axes visible.
CTRL-	<b>ORIGIN POINTS VISIBILITY</b> / Makes the default origin points visible.
CTRL-]	<b>ORIGIN PLANE VISIBILITY</b> / Makes the default origin planes visible.
CTRL-/	<b>ORIGIN AXES VISIBILITY</b> / Makes the default origin axes visible.
CTRL-0	<b>TOGGLE SCREEN</b> / Toggles screen.
CTRL-SHIFT-E	<b>DEGREES OF FREEDOM</b> / Displays degrees of freedom.
CTRL-SHIFT-Q	<b>IMATE GLYPH</b> / Activates the iMate Glyph tool.
CTRL-W	<b>STEERING WHEELS</b> / Displays the Steering Wheel view tool.
END	<b>ZOOM SELECTED</b> / Zooms in on a specifically selected item.
F2	<b>PAN</b> / Views areas of the graphics window. (You must hold the [F2] key down.)
F3	<b>ZOOM SELECTED</b> / Realtime zoom in and out of the graphics window. (You must hold the [F3] key down.)
F4	<b>ROTATE</b> / Rotates objects in the graphics window.

F5	<b>PREVIOUS VIEW</b> / Returns to the last display.
F6	<b>HOME VIEW</b> / Rotates objects in the graphics window into an isometric orientation.
F10	<b>SKETCH VISIBILITY</b> / Makes sketches visible.
HOME	<b>ZOOM ALL</b> / Zooms to the extents of all objects in the graphics window.
PAGE UP	<b>LOOK AT</b> / Looks at a selected entity.
SHIFT-F3	<b>ZOOM WINDOW</b> / Zooms to the area you define with a window.
SHIFT-F5	<b>NEXT VIEW</b> / Advances to the next view.
SHIFT-MIDDLE MOUSE CLICK	<b>ROTATE</b> / Rotates a model.

## Work Features

;	<b>GROUNDING WORK POINT</b> / Creates a grounded work point and activates the 3D Move/Rotate tool.
/	<b>WORK AXIS</b> / Creates a new work axis.
]	<b>WORK PLANE</b> / Creates a new work plane.
.	<b>WORK POINT</b> / Creates a new work point.

# Annotation

<b>BA</b>	<b>AUTO BALLOON</b> / Creates one or more item balloons used to identify components in drawing views.
<b>CAT</b>	<b>CATERPILLAR</b> / Adds a weld caterpillar annotation.
<b>CB</b>	<b>CENTERLINE BISECTOR</b> / Adds a centerline bisector.
<b>CL</b>	<b>CENTERLINE</b> / Adds a centerline.
<b>CM</b>	<b>CENTER MARK</b> / Adds a center mark.
<b>DI</b>	<b>DATUM IDENTIFIER SYMBOL</b> / Adds a datum identifier symbol.
<b>EF</b>	<b>END FILL</b> / Adds a weld end fill annotation in a drawing view or on a 3D weld bead to represent the hatched or filled region indicating the end of a weld bead.
<b>F</b>	<b>FEATURE CONTROL FRAME</b> / Places a feature control frame in drawing files.
<b>FI</b>	<b>FEATURE IDENTIFIER SYMBOL</b> / Adds a feature identifier symbol.
<b>HTF</b>	<b>HOLE TABLE-FEATURES</b> / Selects feature set (all punch center instances from a single feature).
<b>HTS</b>	<b>HOLE TABLE-SELECTION</b> / Selects recovered punch centers individually.
<b>HTV</b>	<b>HOLE TABLE-VIEW</b> / Adds a hole table to a view.
<b>LE</b>	<b>LEADER TEXT</b> / Adds notes with leader lines to a drawing.
<b>PL</b>	<b>PARTS LIST</b> / Creates a parts list.
<b>RT</b>	<b>REVISION TAG</b> / Adds revision tags to a drawing.

<b>RTB</b>	<b>REVISION TABLE</b> / Creates a revision table.
<b>ST</b>	<b>SURFACE TEXTURE SYMBOL</b> / Adds a surface texture symbol.
<b>SY</b>	<b>SYMBOLS</b> / Adds sketched symbols to a drawing sheet.
<b>T</b>	<b>TEXT</b> / Activates the text tool in a drawing file.
<b>TB</b>	<b>TABLE</b> / Creates a table.
<b>WS</b>	<b>WELDING SYMBOL</b> / Provides annotation and acts as a grouping mechanism by referencing multiple beads with a single welding symbol.

<b>PC</b>	<b>PATTERN COMPONENT</b> / Selects one or more components to include in the pattern.
<b>RA</b>	<b>REPLACE ALL</b> / Replaces all occurrences of the component in the current assembly.
<b>V</b>	<b>MOVE COMPONENT</b> / Activates the Move Component command.

# Cable & Harness

<b>AR</b>	<b>AUTOMATIC ROUTE</b> / Automatically routes one or more selected wires into segments based on the shortest path through the network and the segment opening that is closest to the wire start and endpoints.
<b>BR</b>	<b>CHECK BEND RADIUS</b> / Checks for bends that do not meet the specified minimum bend radius on segments, and routed and unrouted wires and cables in the active harness assembly.
<b>BS</b>	<b>BROKEN SKETCH ENTITY</b> / Sets the start point and endpoint for the length to remove from the cable, wire, or segment.
<b>CA</b>	<b>CONNECTOR AUTHORIZING</b> / Adds Cable and Harness-specific data to complete the definition of connectors for use in Cable and Harness.
<b>CC</b>	<b>CREATE CABLE</b> / Creates a cable containing multiple wires that are connected or terminated to selected pins within a harness assembly.

# Assembly

<b>C</b>	<b>CONSTRAINTS</b> / Adds an assembly constraint.
<b>CO</b>	<b>COPY COMPONENTS</b> / Creates a copy of the component.
<b>G</b>	<b>ROTATE COMPONENT</b> / Activates the Rotate tool.
<b>IA</b>	<b>ANALYZE INTERFERENCE</b> / Analyzes assemblies for interference.
<b>MI</b>	<b>MIRROR COMPONENTS</b> / Creates a mirror component.
<b>N</b>	<b>CREATE COMPONENT</b> / Displays the Create In-Place Component dialog box.
<b>P</b>	<b>PLACE COMPONENT</b> / Places a component in the current assembly.

## Cable & Harness

CF	<b>CREATE FOLD</b> / Creates fold.	L	<b>CABLE &amp; HARNESS LIBRARY</b> / Adds new library definitions, and modifies, copies, and deletes existing library definitions for harness objects such as wires, cables, raw ribbon cables, and virtual parts.
CR	<b>CREATE RIBBON CABLE</b> / Creates a ribbon cable between a start and end connector.	NB	<b>NAILBOARD</b> / Specifies the harness assembly and setup to create or edit a nailboard view.
CS	<b>CREATE SEGMENT</b> / Creates a harness segment.	P	<b>PIVOT</b> / Fixes segment points so you can drag the selected segment relative to that fixed point.*
CV	<b>PLACE CONNECTOR VIEWS</b> / Places connector views.	P	<b>PLACE PIN</b> / Places a pin.*
CW	<b>CREATE WIRE</b> / Creates a wire between selected pins within a harness assembly.	PD	<b>PROPERTY DISPLAY</b> / Displays properties for the selected objects.
E	<b>EDIT</b> / Edits nailboard sketch.	PG	<b>PLACE PIN GROUP</b> / Automatically places multiple pins with the specified naming, configuration, and orientation on a part.
EH	<b>EXPORT HARNESS DATA</b> / Exports harness data.	RG	<b>REPORT</b> / Process reports for the active harness assembly.
F	<b>FAN IN</b> / Collapses wire stubs on top of one another to reduce the detail or clutter in nailboard documentation.	RH	<b>REVIEW HARNESS DATA</b> / Reviews harness data.
FO	<b>FAN OUT</b> / Equally distributes wire stubs about the endpoint of a segment using the specified angle and sorting direction.	RT	<b>ROUTE</b> / Routes cables and wires manually.
HA	<b>CREATE HARNESS</b> / Adds the harness subassembly to an assembly file with the specified name and location, and displays the Cable and Harness tab.	RW	<b>RECONNECT WIRE PINS</b> / Reconnects wire pins.
HD	<b>HARNESS DIMENSION</b> / Adds a driven, aligned dimension between 2 points.	SP	<b>CREATE SPLICE</b> / Creates a splice.
HP	<b>HARNESS PROPERTIES</b> / Custom properties of harness components.	TB	<b>TABLE</b> / Creates a table.
IH	<b>IMPORT HARNESS DATA</b> / Imports harness data.	U	<b>UNROUTE</b> / Unroutes selected wires from selected segments.
		V	<b>ASSIGN VIRTUAL PARTS</b> / Assigns virtual parts.

## Dimension

BD	<b>BASELINE DIMENSION</b> / Adds baseline dimensions.
BN	<b>BEND NOTES</b> / Creates or edits a bend note.
CN	<b>CHAMFER NOTE</b> / Adds a chamfer note to a drawing view.
D	<b>GENERAL DIMENSION</b> / Adds a dimension to a sketch or drawing.
HN	<b>HOLE/THREAD NOTES</b> / Adds a hole or thread note with a leader line.
O	<b>ORDINATE DIMENSION SET</b> / Activates the Ordinate Dimension Set command.
OD	<b>ORDINATE DIMENSION</b> / Adds an ordinate dimension.
PN	<b>PUNCH NOTES</b> / Creates a punch note.

\*NOTE: The same alias can perform different commands depending on the design environment and object you are working on.

## Drawing Manager

AV	<b>AUXILIARY VIEW</b> / Places an auxiliary view by projecting from an edge or line in a parent view.
B	<b>BALLOON, BOM</b> / Activates the Balloon tool or displays the BOM Properties dialog box, depending on the work environment.
BV	<b>BASE VIEW</b> / Creates a base view.
BRV	<b>BREAK</b> / Creates a broken, foreshortened view.
BO	<b>BREAK OUT</b> / Removes a defined area of material to expose obscured parts or features in an existing drawing view.
CR	<b>CROP</b> / Use to set boundary type and visibility of crop cut lines for crop operations.
DV	<b>DETAIL VIEW</b> / Provides circular and rectangular shapes of detail profile.
PV	<b>PROJECTED VIEW</b> / Creates a projected view.
RD	<b>RETRIEVE DIMENSIONS</b> / Retrieves model dimensions in a drawing.
SV	<b>SECTION VIEW</b> / Creates a section view.

## Dynamic Simulation

AVI	<b>PUBLISH MOVIE</b> / Creates AVI animation.
CC	<b>CONVERT ASSEMBLY CONSTRAINTS</b> / Converts assembly constraints.
CSA	<b>PUBLISH TO STUDIO</b> / Creates studio animation.
DM	<b>DYNAMIC MOTION</b> / Dynamic part motion.
F	<b>FORCE</b> / Applies force on the selected objects.
FEA	<b>EXPORT TO FEA</b> / Exports to FEA.
J	<b>INSERT JOINT</b> / Inserts joint.
MS	<b>MECHANISM STATUS AND REDUNDANCIES</b> / Provides model status information and guides you through the process of repairing redundant constraints.
OG	<b>OUTPUT GRAPHER</b> / Displays graphs and numerical values of all the input and output variables in a simulation during and after the simulation completes.
RF	<b>REFERENCE FRAME</b> / Creates a Reference Frame from the input provided.
SET	<b>DYNAMIC SIMULATION SETTINGS</b> / Sets options that apply to the entire Dynamic Simulation session.
TO	<b>TORQUE</b> / Applies torque on the selected objects.
TR	<b>TRACE</b> / Creates the absolute or relative kinematic values (trajectory, velocity, and acceleration) of one or more points placed anywhere on a component.
UF	<b>UNKNOWN FORCE</b> / Calculates the required force, torque, or jack to keep a mechanism in static equilibrium for a given position

## Frame Analysis

A	<b>ANIMATE RESULTS</b> / Animates the selected simulation results.
AM	<b>AXIAL MOMENT</b> / Adds an axial moment.
B	<b>PROBE</b> / Displays the simulation results for a specific point on the model.
BC	<b>BOUNDARY CONDITIONS</b> / Toggles display of all boundary conditions.
BD	<b>BEAM DETAIL</b> / Displays detailed results (diagrams, tables) for the selected beams of a structure.
BM	<b>BENDING MOMENT</b> / Applies the bending moment to the selected beam.
C	<b>CREATE SIMULATION</b> / Creates a simulation.
CB	<b>COLOR BAR</b> / Edits color bar.
CL	<b>CONTINUOUS LOAD</b> / Applies and distributes uniform load along the beam you select.
D	<b>DIAGRAM</b> / Adds custom user diagram.
EX	<b>EXPORT</b> / Exports to robot structural analysis.
F	<b>FORCE</b> / Applies a force of the specified magnitude to the selected faces, edges, or vertices.
FL	<b>FLOATING PINNED CONSTRAINT</b> / Adds floating pinned constraint.
LB	<b>BEAM LABELS</b> / Toggles display of beam labels.

## Frame Analysis

LN	<b>NODE LABELS</b> / Toggles display of node labels.
LP	<b>PROBE LABELS</b> / Toggles visibility of probe labels.
LS	<b>LOCAL SYSTEMS</b> / Toggles display of beam local systems.
LV	<b>LOAD VALUES</b> / Toggles display of load values.
MA	<b>BEAM MATERIALS</b> / Enables you to change the material of the beam.
MO	<b>MOMENT</b> / Applies the general moment to the selected beam.
N	<b>NO SHADING</b> / Turns off the Shaded Results display.
P	<b>BEAM PROPERTIES</b> / Provides physical and engineering properties of a frame member, and enables you to customize the data.
PN	<b>PINNED CONSTRAINT</b> / Applies pinned constraint when no displacement and free rotation is allowed on beam or node.
R	<b>REPORT</b> / Generates a report.
RE	<b>RELEASE</b> / Assigns a release of specified degrees of freedom to a selected beam in a structure.
RL	<b>RIGID LINK</b> / Enables you to define a rigid link between selected nodes in a frame structure.
S	<b>SIMULATE</b> / Runs a simulation.
X	<b>FIXED CONSTRAINT</b> / Applies a fixed constraint on selected faces, edges, or vertices.

## Mold Design

2D	<b>2D DRAWING</b> / Creates one or more drawing documents.	CH	<b>CREATE HEEL</b> / Creates one or more heels on an insert.*
AM	<b>ANIMATE RESULTS</b> / Animates the selected simulation results.	CI	<b>CREATE INSERT</b> / Creates an insert.
AO	<b>ADJUST ORIENTATION</b> / Prompts the Adjust Orientation dialog box.	CM	<b>COMBINE CORES AND CAVITIES</b> / Creates, edits, or deletes a combined core or cavity.
AP	<b>ADJUST POSITION</b> / Prompts the Adjust Position dialog box.	CP	<b>PLACE CORE PIN</b> / Places a core pin into a mold.
AR	<b>AUTO RUNNER SKETCH</b> / Creates an auto runner sketch.	CS	<b>COOLING COMPONENT</b> / Specifies the components that conduct the coolant through the cooling channels.
AU	<b>MOLD BASE AUTHOR</b> / Assigns component attributes before inserting a user defined mold base.	CW	<b>COLD WELL</b> / Defines the cold wells that are at the end of the sprue or on a runner.
BC	<b>BRIDGE CURVE</b> / Connects curves by G2 continuity.	DW	<b>DEFINE WORKPIECE SETTING</b> / Enables you to generate a rectangular or cylindrical workpiece, which is the precondition for generating the core and cavity.
BL	<b>MOLD BOOLEAN</b> / Integrates the interference checking function and the Boolean calculation.	EJ	<b>EJECTOR</b> / Creates an ejector pin to eject a plastic part.
BR	<b>BOUNDED RUNOFF SURFACE</b> / Creates a runoff surface.	EM	<b>EDIT MOLDABLE PART</b> / Edits a moldable part.
CC	<b>COOLING CHANNEL CHECK</b> / Checks cooling channels for safety, interference, and distance between cooling system segments.*	ES	<b>USE EXISTING SURFACE</b> / Creates, edits, or deletes the existing patching or runoff surface.
CC	<b>PLACE CORE AND CAVITY</b> / Places or deletes imported core and cavity.*		
CD	<b>CORE/CAVITY</b> / Core/Cavity design and analysis environment.		
CH	<b>COOLING CHANNEL</b> / Creates a cooling channel.*		

\*NOTE: The same alias can perform different commands depending on the design environment and object you are working on.

# Mold Design

ET	<b>EXTEND RUNOFF SURFACE</b> / Extends runoff surface.	PH	<b>CREATE PLANAR PITCH</b> / Creates planar pitch.	SM	<b>SELECT MATERIAL</b> / Selects material.
F	<b>FINISH CORE/CAVITY</b> / Activates core and cavity environment.	PI	<b>PLACE INSERT</b> / Places a previously generated insert in the mold assembly.	SP	<b>SECONDARY SPRUE</b> / Provides the means to add a new secondary sprue to the mold design and to edit, delete, and adjust the dimensions of the secondary sprue.
GC	<b>GENERATE CORE AND CAVITY</b> / Creates the required files for the core and cavity.	PP	<b>PLASTIC PART</b> / Places one or more parts into a Mold Design project.	UM	<b>USER MOLD BASE</b> / Places a user-defined mold base.
GP	<b>GATE LOCATION</b> / Sets gate locations.	PR	<b>SET PLOT PROPERTY</b> / Sets plot property.	WP	<b>WORKPIECE POCKET</b> / Creates pockets for a rectangular workpiece.
GT	<b>GATE</b> / Adds a gate.	PS	<b>CREATE PATCHING SURFACE</b> / Creates patching surface.*		
JM	<b>JOB MANAGER</b> / Monitors the progress of analyses and, if necessary, stops an analysis.	PS	<b>MOLD PROCESS SETTINGS</b> / Opens mold process settings.*		
LA	<b>LIFTER</b> / Places a lifter assembly into the mold so the mold can open and close smoothly.	PT	<b>PATTERN</b> / Duplicates one plastic part and arranges the resulting occurrences in a rectangular, circular, or variable pattern.		
LR	<b>LOCATING RING</b> / Creates a locating ring.	RN	<b>RUNNER</b> / Creates a runner.		
MB	<b>MOLD BASE</b> / Provides the tools to create a mold base.	RR	<b>RADIATE RUNOFF SURFACE</b> / Creates, edits, or deletes a radiated runoff surface.		
MF	<b>MOLD FILL ANALYSIS</b> / Predicts the thermoplastic polymer flow inside the mold during the filling phase.	RS	<b>CREATE RUNOFF SURFACE</b> / Creates a runoff surface.		
MP	<b>PART PROCESS SETTINGS</b> / Determines the optimum mold temperature, melt temperature, and injection time based on the selected material and part geometry.	SA	<b>SLIDER</b> / Creates a standard slider.		
MS	<b>MANUAL SKETCH</b> / Activates the sketch environment of a part where you can draw an insert sketch.	SB	<b>SPRUE BUSHING</b> / Creates a custom or vendor sprue bushing.		
NM	<b>CREATE MOLD DESIGN</b> / Creates mold design.	SK	<b>MOLD SHRINKAGE</b> / Estimates the percentage of shrinkage that occurs based on the mold dimensions.*		
PF	<b>PART FILL ANALYSIS</b> / Reveals information about the flow of polymer through parts.	SK	<b>PART SHRINKAGE</b> / Estimates the percentage of shrinkage on an individual part.*		
		SL	<b>LOCK SET</b> / Creates a side lock or an interlock.		

## Part

1	<b>TOGGLE SMOOTH</b> / Activates the toggle smooth command.
E	<b>EDIT FORM</b> / Edits an existing form to add, remove, or modify controls.

\*NOTE: The same alias can perform different commands depending on the design environment and object you are working on.

## Placed Features

CH	<b>CHAMFER</b> / Creates a chamfer.
DE	<b>DIRECT EDIT</b> / Allows you to make quick edits to existing solids.
EF	<b>END FILL</b> / Adds a weld end fill annotation in a drawing view or on a 3D weld bead to represent the hatched or filled region indicating the end of a weld bead.
F	<b>FILLET</b> / Creates a fillet in part and assembly files.
GW	<b>GROOVE WELD</b> / Creates a groove weld feature in a weldment assembly that connects two face sets with a solid weld bead.
MI	<b>MIRROR</b> / Creates a mirror feature.
Q	<b>CREATE IMATE</b> / Displays the Create iMate dialog box, for developing iMates.
RP	<b>RECTANGULAR PATTERN</b> / Creates rectangular pattern of feature.
S	<b>2D SKETCH</b> / Activates the 2D Sketch command.
S3	<b>NEW SKETCH</b> / Activates the 3D Sketch command.
SH	<b>SHELL</b> / Removes material from a part interior, creating a hollow cavity with walls of a specified thickness.
TH	<b>THREAD</b> / Creates threads in holes or on shafts, studs, or bolts.
W	<b>FILLET</b> / Activates the Fillet tool.
WS	<b>WELDING SYMBOL</b> / Provides annotation and acts as a grouping mechanism by referencing multiple beads with a single welding symbol.

## Presentation

CV	<b>CREATE VIEW</b> / Adds a new presentation view.
T	<b>TWEAK COMPONENTS</b> / Tweaks components in a presentation.

## Sheet Metal

BE	<b>BEND</b> / Creates a bend.
CC	<b>CORNER CHAMFER</b> / Activates the corner chamfer command.
CF	<b>CONTOUR FLANGE</b> / Creates a contour flange.
CR	<b>CORNER ROUND</b> / Adds fillets or rounds to one or more corners of a sheet metal part.
CS	<b>CORNER SEAM</b> / Defines the corner seams you add to sheet metal faces.
FA	<b>FACE</b> / Creates a sheet metal face.
FO	<b>FOLD</b> / Defines the fold in a sheet metal face along a sketched line that terminates at face edges.
HEM	<b>HEM</b> / Creates a folded hem along sheet metal edges.
PT	<b>PUNCHTOOL</b> / Defines the punch of a 3D shape into a sheet metal face.
ST	<b>SHEET METAL DEFAULTS</b> / Changes the options and parameters for the active sheet metal part from those options defined by the selected Sheet Metal Rule.

## Sketch

A	<b>BASELINE SET, ANIMATE, ARC</b> / Activates the Baseline Set, Animate, or Arc tool, depending on the work environment.
A3	<b>THREE POINT ARC</b> / Draws a three-point arc using dynamic input.
AD	<b>AUTO DIMENSION</b> / Prompts the Auto Dimension dialog box.
BE	<b>BEND</b> / Creates a bend.
CE	<b>CENTER POINT CIRCLE</b> / Draws a center point circle using dynamic input.
CH	<b>CHAMFER</b> / Creates a chamfer.
CP	<b>CIRCULAR PATTERN</b> / Creates circular pattern of sketch geometry.
EL	<b>ELLIPSE</b> / Creates an ellipse with a center point, a major axis, and a minor axis that you define.
EX	<b>EXTEND</b> / Makes surfaces larger in one or more directions.
FI	<b>FILLET</b> / Adds fillets or rounds to one or more edges of a part, between two face sets, or between three adjacent face sets.
H	<b>FILL/HATCH SKETCH REGION</b> / Hatches or color fills an enclosed boundary in a drawing sketch.
I	<b>VERTICAL</b> / Activates the vertical constraint tool.
L	<b>LINE</b> / Creates a line or arc.
MI	<b>MIRROR</b> / Mirrors sketch geometry about a selected plane.

## Sketch

<b>MO</b>	<b>MOVE</b> / Moves selected sketch geometry from point to point, or move a copy of the geometry.
<b>O</b>	<b>OFFSET</b> / Duplicates selected sketch geometry, and positions it an offset distance from the original.
<b>PO</b>	<b>POINT, CENTER POINT</b> / Sketches points and center points.
<b>POL</b>	<b>POLYGON</b> / Creates polygon shapes.
<b>REC</b>	<b>TWO POINT RECTANGLE</b> / Draws a two-point rectangle using dynamic input.
<b>REC3</b>	<b>THREE POINT RECTANGLE</b> / Draws a three-point rectangle using dynamic input.
<b>RO</b>	<b>ROTATE COMPONENT</b> / Activates the Rotate Component command.
<b>RP</b>	<b>RECTANGULAR PATTERN</b> / Creates rectangular pattern of sketch geometry.
<b>SP</b>	<b>SPLINE</b> / Creates a spline.
<b>TC</b>	<b>TANGENT CIRCLE</b> / Creates circle tangent to three lines.
<b>X</b>	<b>TRIM</b> / Activates the Trim command.

## Sketch Features

<b>BP</b>	<b>BEND PART</b> / Bends a portion of a part.
<b>E</b>	<b>EXTRUDE</b> / Extrudes a profile.
<b>H</b>	<b>HOLE</b> / Creates a hole in a feature.
<b>LO</b>	<b>LOFT</b> / Creates lofted features or bodies by blending multiple profiles into smooth shapes between the profiles or part faces.
<b>R</b>	<b>REVOLVE</b> / Revolves a sketch.
<b>SW</b>	<b>SWEEP</b> / Creates a sweep feature.

## Stress Analysis

<b>A</b>	<b>ANIMATE RESULTS</b> / Animates the selected simulation results.
<b>AC</b>	<b>AUTOMATIC CONTACTS</b> / Compares automatic contacts.
<b>BC</b>	<b>BOUNDARY CONDITIONS</b> / Toggles display of all boundary conditions.
<b>BE</b>	<b>BEARING LOAD</b> / Applies a bearing load to a full or partial cylindrical surface.
<b>BO</b>	<b>BODY LOAD</b> / Defines angular velocity or angular or linear acceleration for the model.
<b>CC</b>	<b>COLOR BAR</b> / Displays the Color Bar settings dialog box where you adjust the color bar display parameters.
<b>CF</b>	<b>FIXED CONSTRAINT</b> / Applies a fixed constraint on selected faces, edges, or vertices.

<b>CP</b>	<b>CONVERGENCE PLOT</b> / Displays the plot within a dialog box.*
<b>CP</b>	<b>PIN CONSTRAINT</b> / Sets the pinned constraint origin and offset.*
<b>CS</b>	<b>FRICTIONLESS CONSTRAINT</b> / Applies a frictionless constraint on selected faces.
<b>CT</b>	<b>CONTOUR SHADING</b> / Displays color changes using a strict banding between colors.
<b>FO</b>	<b>FORCE LOAD</b> / Applies a force of the specified magnitude to the selected faces, edges, or vertices.
<b>GR</b>	<b>GRAVITY LOAD</b> / Creates a gravity load.
<b>MAT</b>	<b>ASSIGN MATERIALS</b> / Opens the assign materials dialogue box to assign materials to a component.
<b>MAX</b>	<b>MAXIMUM RESULT</b> / Turns on and off the display of the point of maximum result in the mode.
<b>MC</b>	<b>MANUAL CONTACT</b> / Adds manual contact conditions to selected geometry elements.
<b>MIN</b>	<b>MINIMUM RESULT</b> / Turns on and off the display of the point of minimum result in the model.
<b>ML</b>	<b>LOCAL MESH CONTROL</b> / Adds a local mesh control.
<b>MO</b>	<b>MOMENT LOAD</b> / Applies a load of the specified magnitude around the axis and perpendicular to the face.
<b>MS</b>	<b>MESH SETTINGS</b> / Specifies the mesh sizing and coarseness.

\*NOTE: The same alias can perform different commands depending on the design environment and object you are working on.

## Stress Analysis

MV	<b>MESH VIEW</b> / Displays the element mesh used in the solution with the result contours. Also displays the mesh over the undeformed model.
N	<b>CREATE SIMULATION</b> / Creates new simulation.
NP	<b>NO SHADING</b> / Turns off the Shaded Results display.
P	<b>PROBE</b> / Activates the Probe command. You place probes as needed in areas of interest to display the stress values for that point.
PL	<b>PROBE LABELS</b> / Toggles the visibility of probe labels.
PR	<b>PRESSURE LOAD</b> / Pressure load.
R	<b>REPORT</b> / Generates a report.
RF	<b>REMOTE FORCE LOAD</b> / Applies a force of the specified magnitude to the selected face.
S	<b>SIMULATE</b> / Runs a simulation.
SC	<b>RESULT DISPLACEMENT SCALE</b> / Adjusts displacement scale.
SET	<b>STRESS ANALYSIS SETTINGS</b> / Activates the stress analysis settings dialog box.
SM	<b>SMOOTH SHADING</b> / Displays color changes using a blended transition.
SS	<b>SAME SCALE</b> / Maintains the same scale while viewing different results.
T	<b>PARAMETRIC TABLE</b> / Accesses the parametric table to specify design constraints and parameter ranges.

## Tools

LA	<b>EDIT LAYERS</b> / Specifies a layer name and attributes for common display properties of drawing objects.
M	<b>MEASURE DISTANCE</b> / Opens the Measure Distance window.
OP	<b>APPLICATION OPTIONS</b> / Opens the Application Options settings.
SE	<b>EDIT STYLES</b> / Style and Standard Editor.

## Tube & Pipe

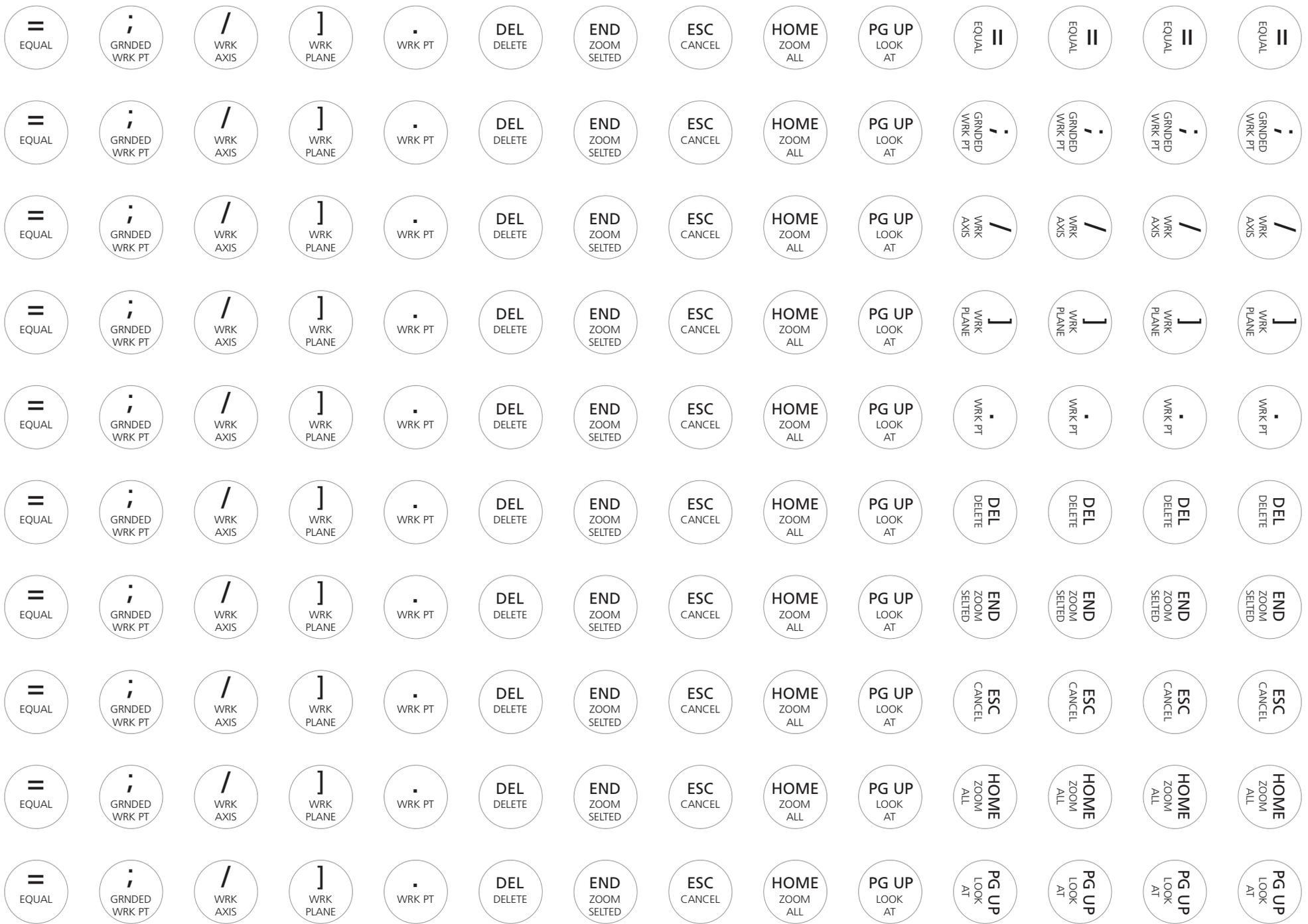
AT	<b>TUBE &amp; PIPE AUTHORIZING</b> / Authors an iPart or a normal part for publishing to the Content Center Library.
CF	<b>CONNECT FITTINGS</b> / Connects existing fittings.
DR	<b>DERIVED MOTION</b> / Creates or edits a derived route.
ES	<b>EDIT BASE SKETCH</b> / Edits the base 3D sketch.
HL	<b>HOSE LENGTH</b> / Edits hose length by changing end tangency weights.
IG	<b>INCLUDE GEOMETRY</b> / Introduces reference geometry to the route sketch.
IN	<b>INSERT NODE</b> / Inserts route node into segment.
IO	<b>ISOGEN OUTPUT</b> / Saves a file with a specified name and ISOGEN file type.

MN	<b>MOVE NODE</b> / Drags node to a new location.
MS	<b>MOVE SEGMENT</b> / Drags segment to a new location.
NR	<b>NEW ROUTE</b> / Adds new rigid piping and bent tubing routes to a pipe run.
PF	<b>PLACE FITTING</b> / Picks up a fitting from your project work space to place into the active run.
PR	<b>CREATE PIPE RUN</b> / Creates pipe run.*
PR	<b>POPULATE ROUTE</b> / Populates route.*
RT	<b>ROUTE</b> / Starts or continues route definition.
ST	<b>TUBE AND PIPE STYLES</b> / Copies and modifies existing style definitions and delete styles you no longer use.
WT	<b>GROUNDING WORK POINT</b> / Grounded Work Point.

## View

ORBIT	<b>FREE ORBIT</b> / Rotates a model about the center of the screen or about axes in model space.
PAN	<b>PAN</b> / Pans the view.
VC	<b>VIEWCUBE</b> / Toggles the view of the ViewCube.
Z	<b>ZOOM WINDOW</b> / Zooms in on a specified area.

\*NOTE: The same alias can perform different commands depending on the design environment and object you are working on.



PRINTABLE KEYBOARD STICKERS [14 COMPLETE SETS]  
FOR USE WITH: <http://www.onlinelabels.com/OL32.htm>