Rhino Training - Research Computing Services
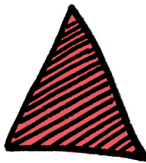
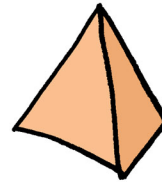| Command	Shortcut	Result	Learning Objective
-	Left Click	Select, click and drag to select multiple	navigation
-	Right Click	Pan around centre mass; repeat last command, ""enter"	
-	Ctrl + Shift	Select individual face or part	
ZE	-	Zoom Extent: Centre rotation around all objects	
ZS	-	Zoom Selected: Centres rotation around selected object	
Move	-	Move surface from a selected point	
-	Enter	Repeat last command; progress command line	selecting shapes
-	Spacebar	Repeat last command; progress command line	
Group	Ctrl + G	Group parts together without joining	
Ungroup	-	Remove group status	
BooleanDifference	-	Removing the shape of one object from the mass of another, Tinkercad void functions	trimming shapes
BooleanIntersection	-	Two overlapping objects are removed, leaving the overlapping section behind	
BooleanUnion	-	Joins two polysurfaces together, removing any overlap and forming a closed surface	
Cap	-	Fills in a simple hole within a surface; usually needs to be planar (flat)	drawing shapes
Curve	-	Draw a curve through a series of points	
ExtrudeCrv	-	Extrudes the shape of a curve in one or both directions	
ExtrudeSrf	-	Extrudes a surface in one or both directions	
Join	Ctrl + J	Join touching components into a surface	
Loft	-	Connect multiple lines, or curves through a single approximated surface	
Mirror	-	Symmetrical copy from a set point	
Offset	-	Copies a curve to an adjacent angle at a set distance	
OffsetSrf	-	Adds thickness to an object by creating a larger copy around it, keeping the same proportions, but adding a second skin.	
Planarsrf	-	Fills a curve with a solid surface	
Polyline	-	Draw a connected series of straight lines	

definitions

Curve   Refers to lines. Any drawn lines, any drawn shapes closed or open

Planar surface   A single face that is solid or filled.

Poly surface   A collection of planar surfaces

Boolean   The term for intersecting shapes and how the intersection is handled.

curve   planar surface   poly surface