

Media 1 Rhino Tutorial

Camera Movement and Selection Tools:

Holding Rclick = Moving camera around in a spherical manner

Holding Shift + Rclick = Moving Camera on an axis (Vertical and Horizontal)

Holding Ctrl + Rclick = Zooming In and out quickly

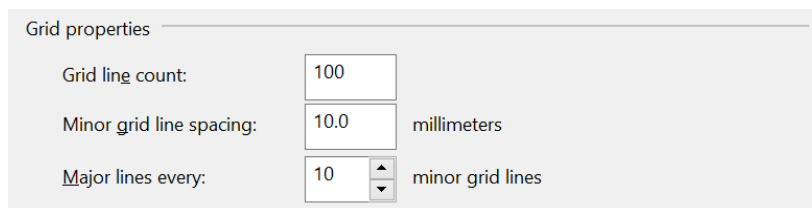
Holding Alt + Rclick = Zooming In and out slower

Setting up a Rhino File:

Click File -> Properties -> Specify the UNITS & GRIDS to the scale you need your project to be.

For this specific tutorial since our project is quite small, try to set it in **millimetres** (mm).

Grid – More of a personal setting, but if you are confused, you can use the settings below.



You can use the **distance** command to confirm you have the right units.

COMMANDS:

Check out the commands shown on the left side of Rhino when you first open it up.

Measurement Commands -

Distance, Length, Angle,

Common Commands -

SnapSize, Planarsrf, Sweep1, Hide, Show & ShowSelected, Lock & Unlock, Offset, Join, Split, Rotate & Rotate3D, DupEdge, ZSA.

- Scale (3 Types; Scale, Scale1d, Scale2d)
- Group (3 types; Group, Ungroup, RemoveFromGroup),

3D Commands (The fun commands) -

ExtrudeCrv, Loft, Cap, CageEdit, Pipe, Cap, Shell

- Boolean (Mainly 3 types; BooleanUnion, Boolean Split, Boolean Difference)

Printing Commands -

- Unrollsurf, Make2d

NOTE;

This pdf is made just to remind you of the commands gone through the tutorial, if you are unsure what the command or a tool does, you can always check out the Official McNeel.com website or search it up on Youtube.