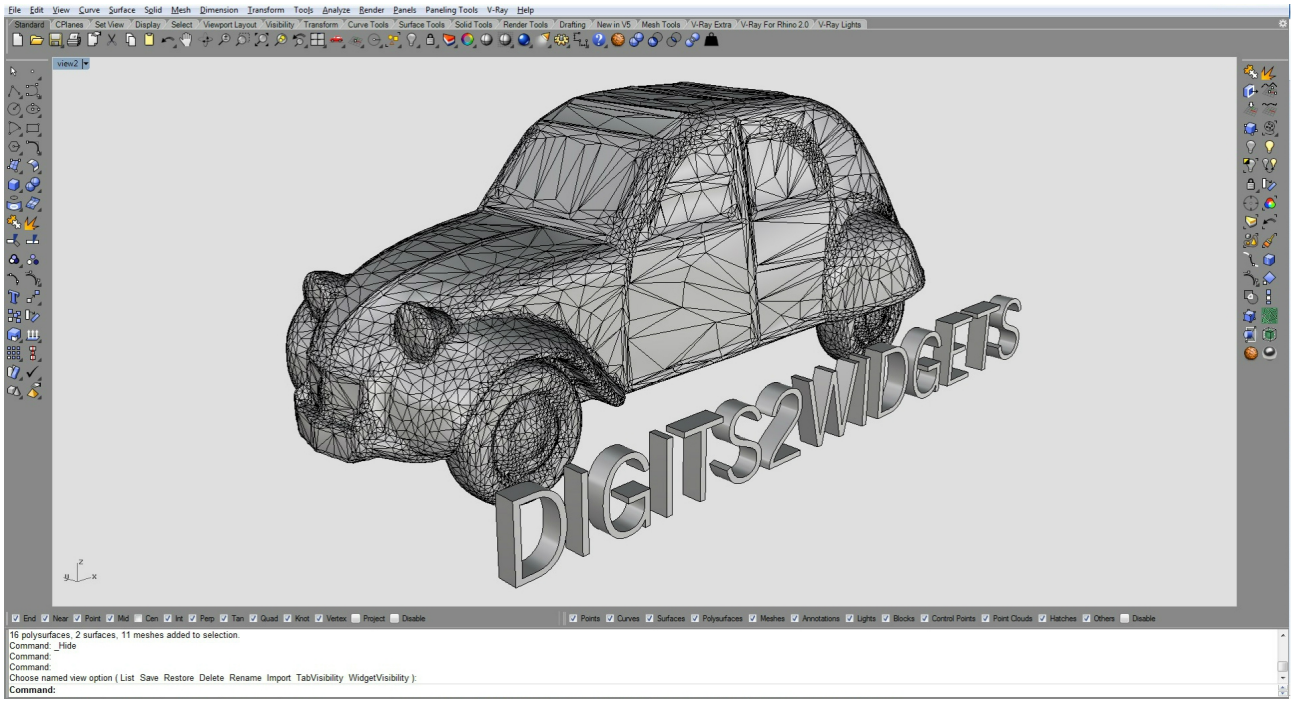
Preparing Files For 3d Printing With Rhino V5

Contents

General Principles

Rhino 3D is a great tool to produce clean files for 3D printing, but it is very important to remember that not all objects that are created within Rhino can automatically be successfully 3D printed.
The final product of your design is an .stl file that is then sent to us for printing. This file contains triangulated geometry – a "mesh" that can be read by any 3D printer.



Only closed surfaces, closed polysurfaces and closed meshes will export correctly to an .stl file.

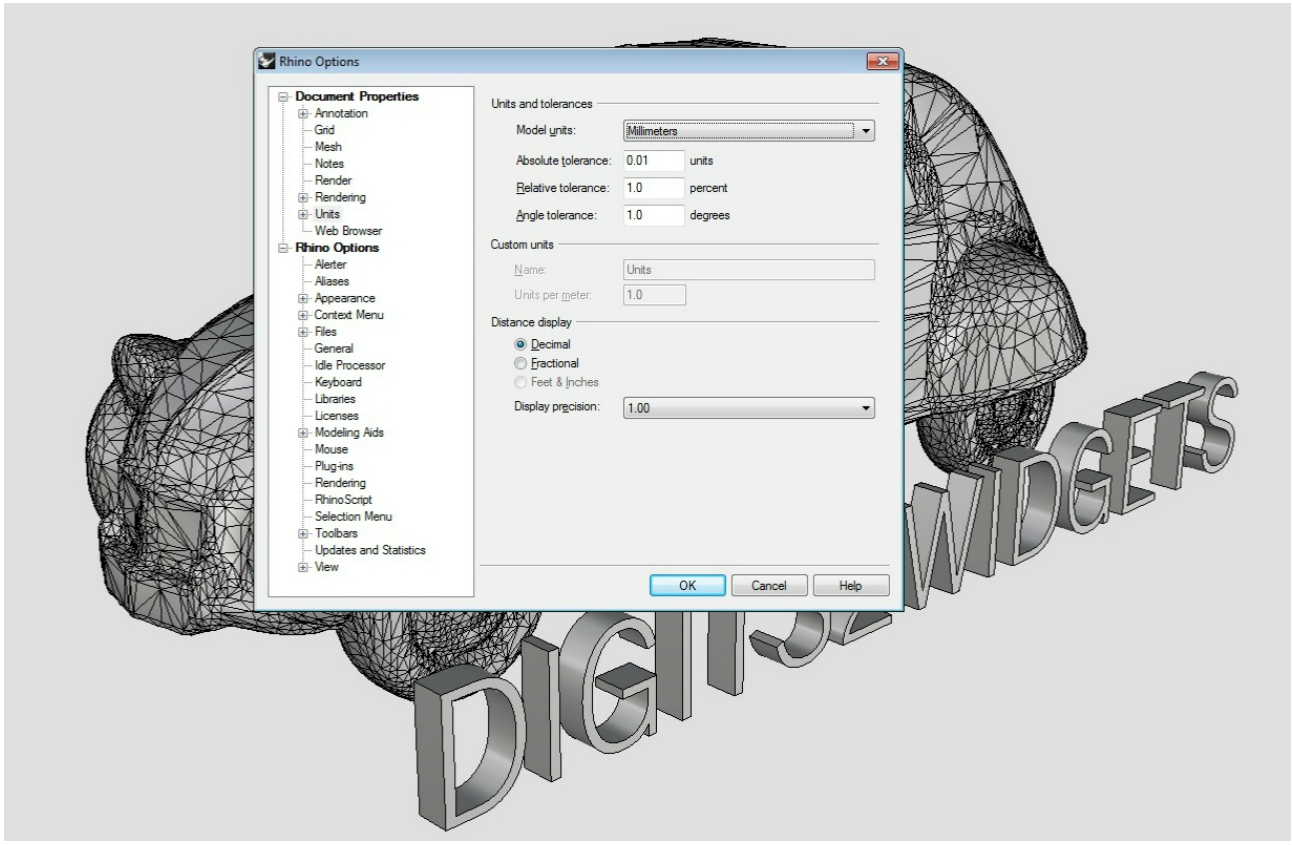
Document Set Up

It is a good idea to prepare your drawings in the scale that you want to print them. This is especially important with Architectural models where the general tendency is to design at 1:1 scale and then simply export it at a smaller scale for printing. This approach causes many problems, mainly where far too fine a detail is expected for any 3D printing process.

Lets start with setting Rhino up. There are 3 main areas to focus on:

1. **units and tolerances** will allow better control during the design process and easy export to .stl
2. **display options** can be set in a way that will allow you to spot any potential problems during design process
3. additional **toolbars** can be loaded to allow access to .stl specific commands

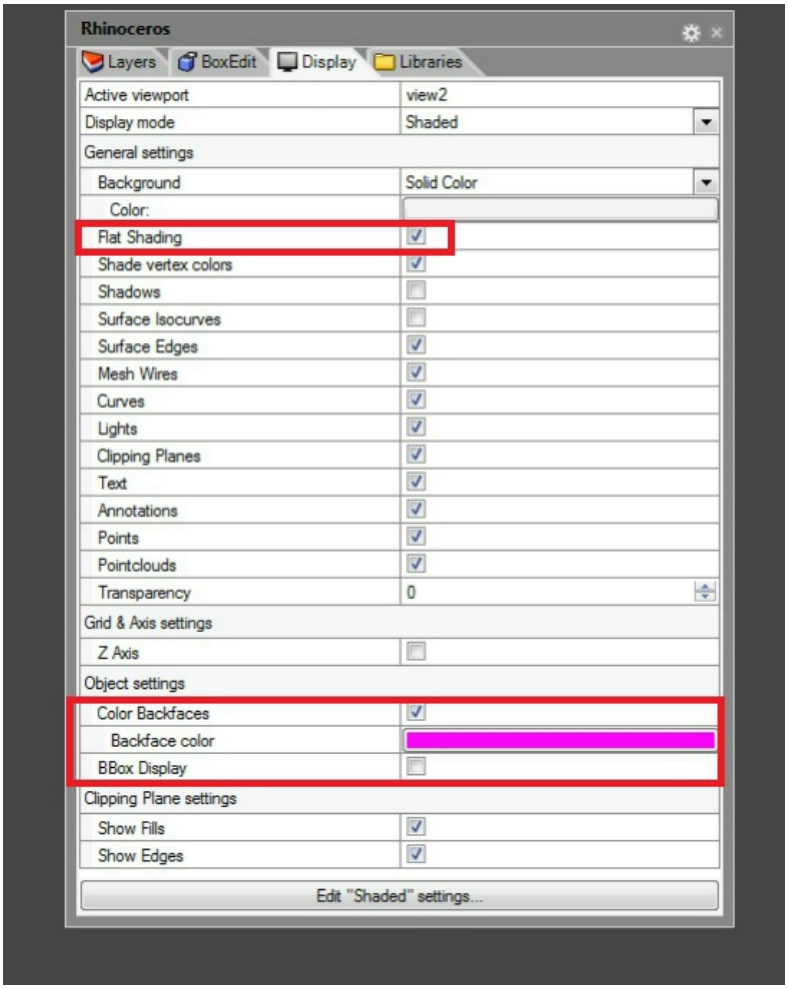
In order to set the units and tolerances in Rhino you need to open **Options** by typing "options" or through menu > options



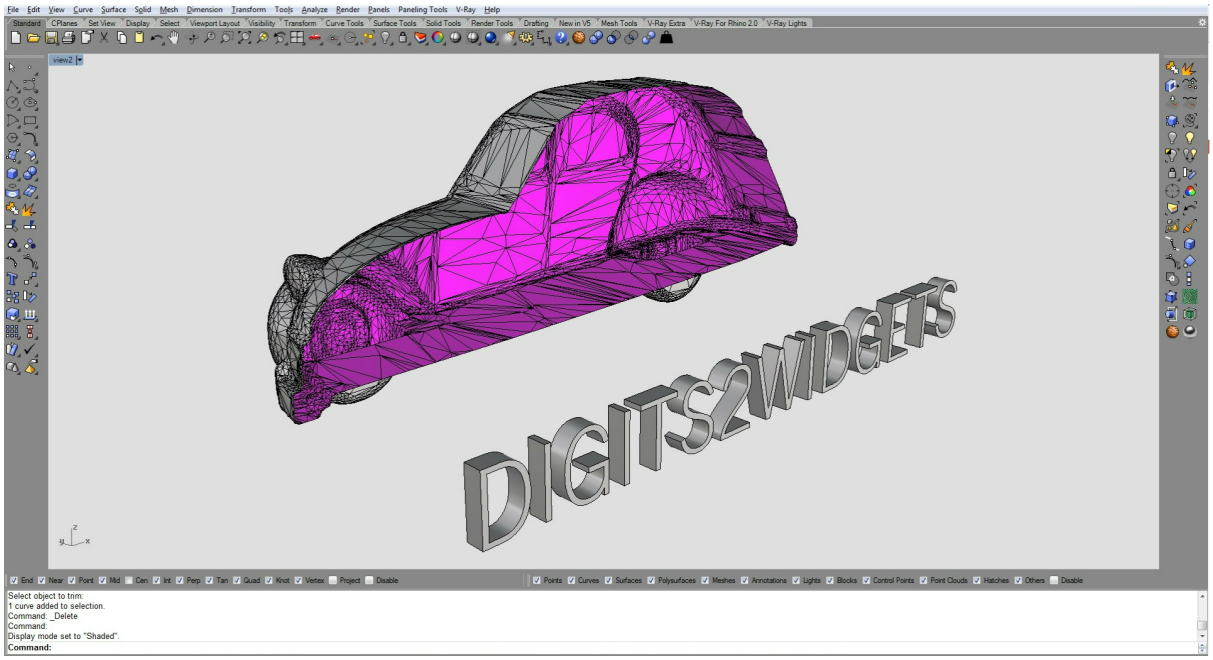
Model units – Millimetres
Absolute tolerance – 0.01

These settings will be perfect for most applications, but if the final drawing has extremely small detail, absolute tolerance may need to be increased.

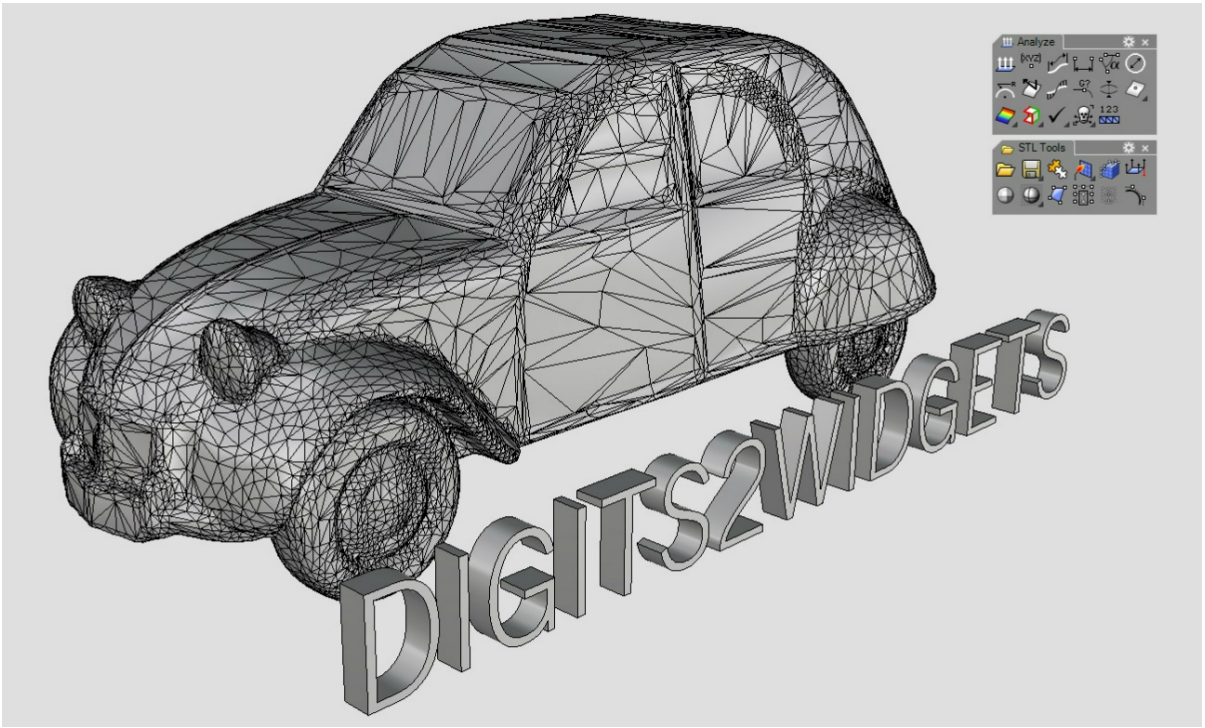
It is a good practice to tweak certain display settings like “colour backfaces” and “flat shading”, although the last one may considerably slow the computer down depending on the complexity of the design.



Once enabled, the back faces are visible in the viewport



Bringing out some additional toolbars like **Analyze** and **STL Tools** will allow quick access to .stl specific commands. These toolbars can be customized.



Basic Drawing Rules

To avoid bad geometry it is a good idea to enable the **CheckNewObjects** option at the very beginning of the design process. This will force Rhino to analyse every new object that is created and will give the opportunity to fix any issues as they arise.

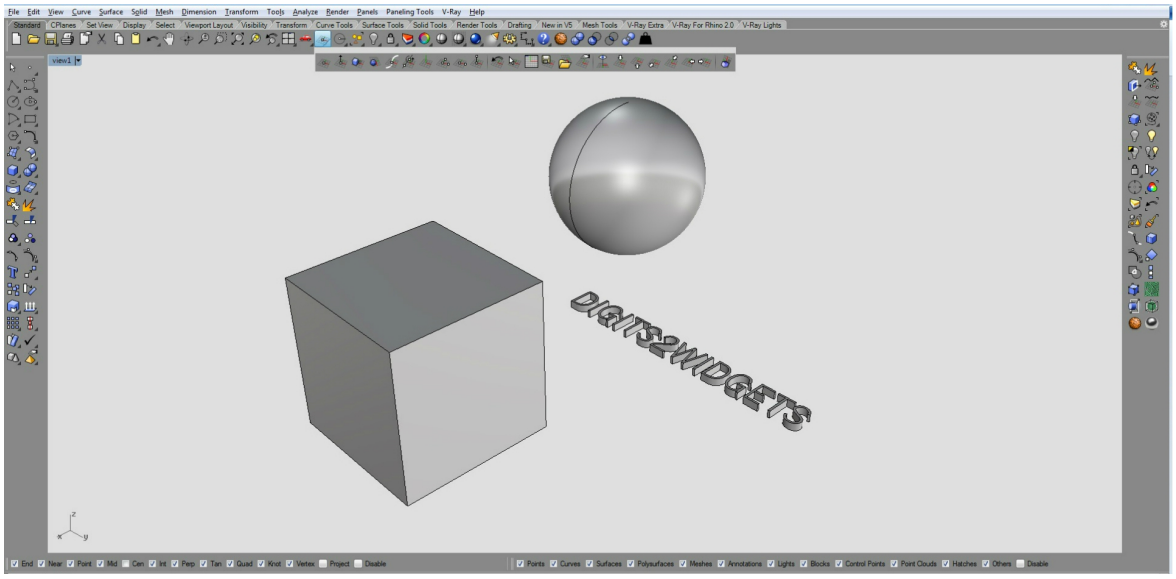
The right use of **Osnap** is very helpful in correct modelling.

Any extrusions need to be made from closed and planar curves to create closed objects.

Avoid trimming polysurfaces, and use boolean operations instead, this ensures that there are no open polysurfaces left after the operation. A good alternative to trimming is the **WireCut** command.

Also try to avoid **Patch** command as it is very tricky to get good result.

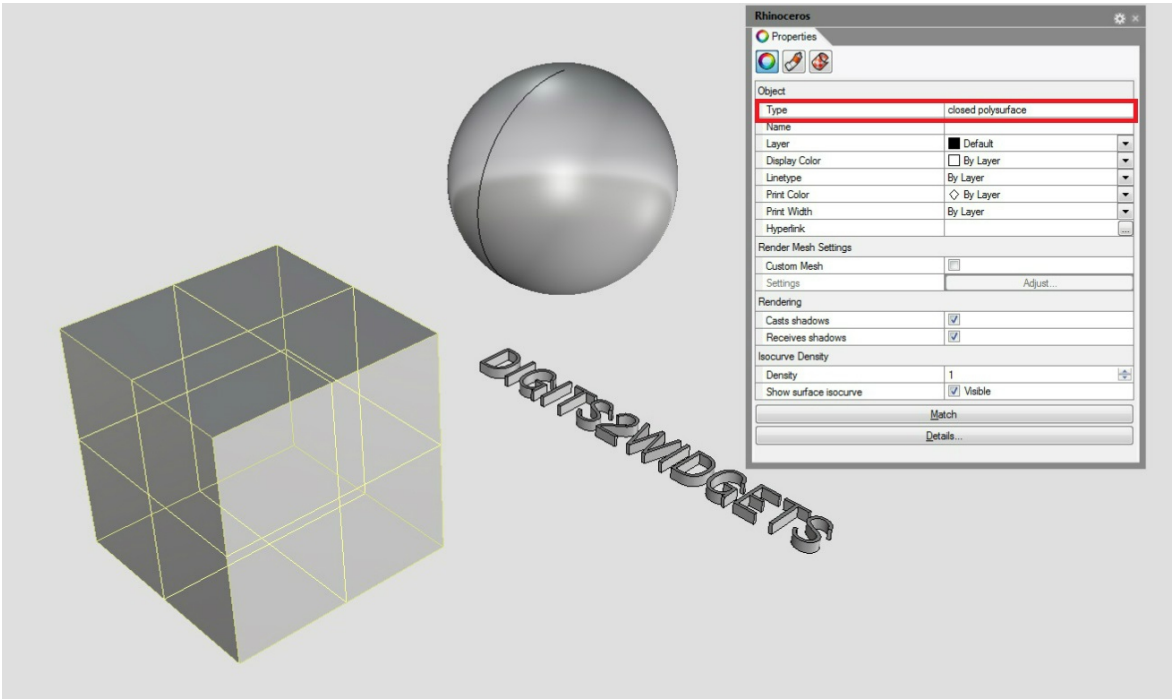
Depending on the geometry it is advisable to use extensively different construction planes **CPlane**



How To Find And Fix Problems

Once the design is complete or imported into Rhino it is time to verify for "printability". This will ultimately avoid additional costs for file fixing.

The first step is to check the object in **Properties** panel. Closed surfaces, closed polysurfaces and closed meshes will be exported correctly to .stl file.



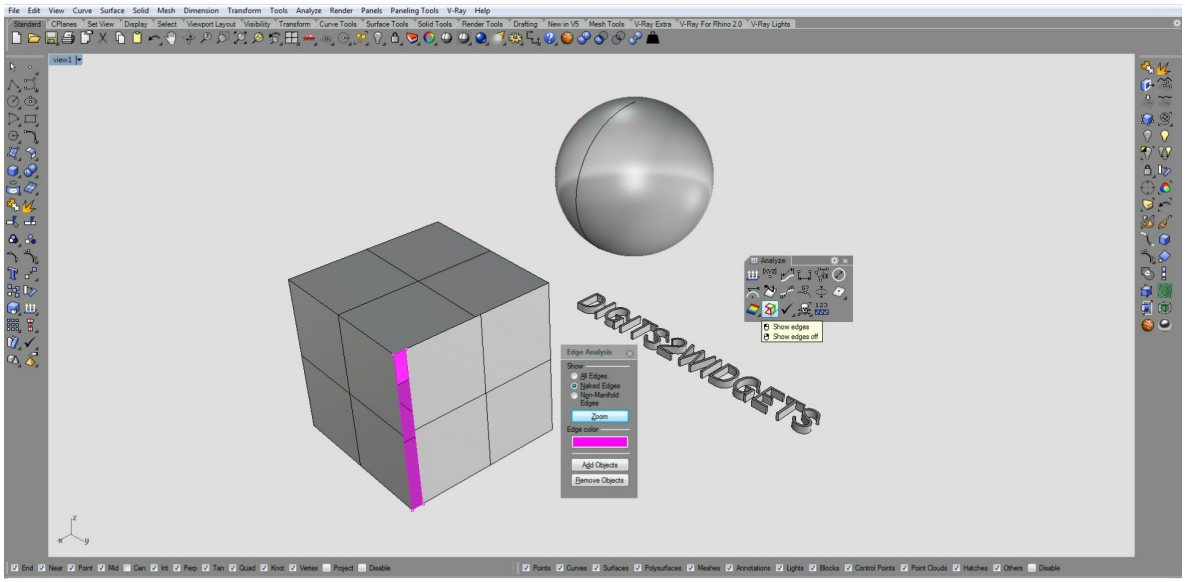
Alternatively **SelBadObjects** command selects all the bad objects at one go.

To fix the issues it may be necessary to go through the "bad objects" one by one.

One of the most common problems is “naked edges”. What it means is that the object in not closed and some of the edges are not touching (are not joined with) other edges.

Lets make one “bad” object for illustration purposes.

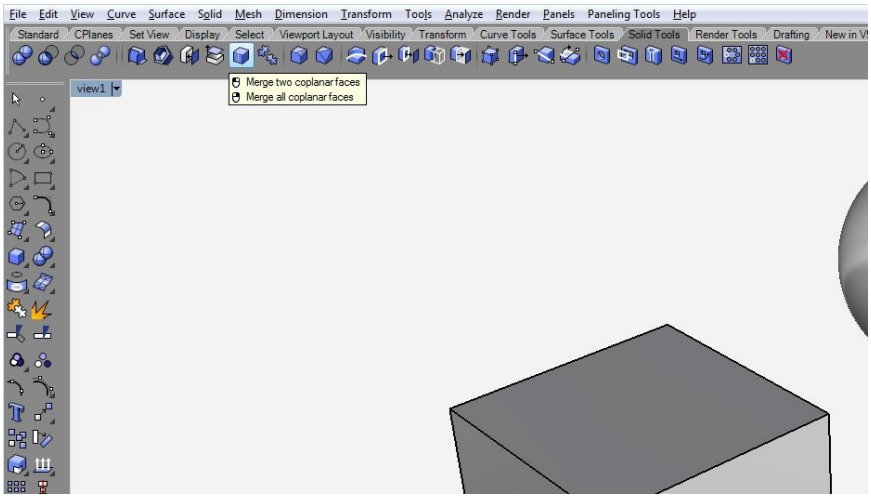
To find “naked edges” select the object(s) and use the **Showedges** command. From menu go to analyze>edge tools>show edges or use the **Analyze** toolbar. Although thanks to display settings it is very easy to spot the issues.



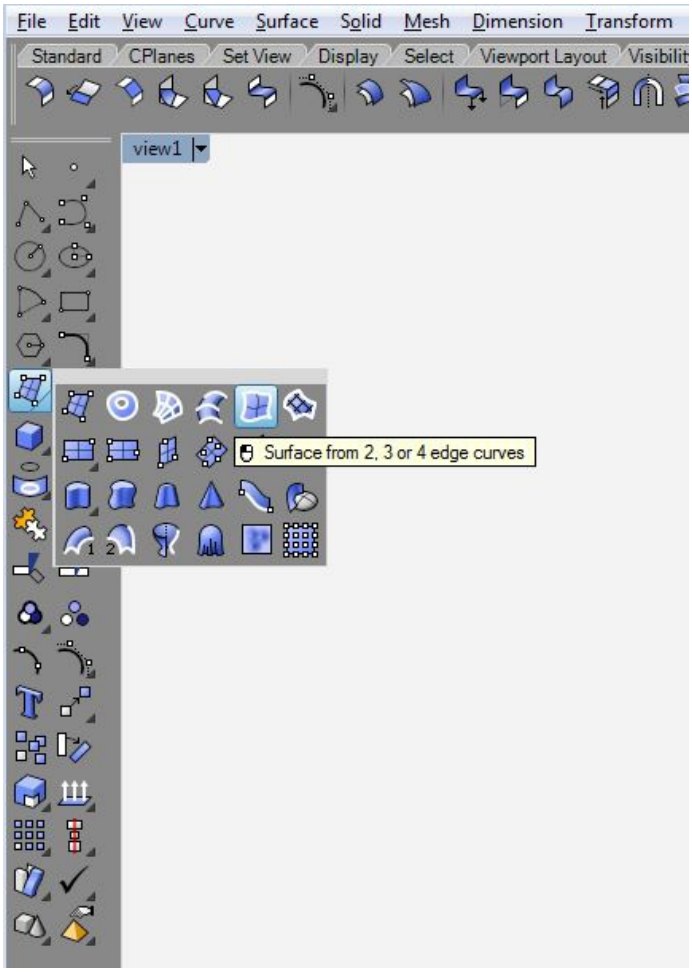
When “show naked edge” is selected Rhino will mark all “naked edges”.

Once the problem has been identified it needs to be fixed. There are few ways of dealing with naked edges, depending on the complexity of geometry.

In this example easiest would be to use command **Cap**. This command works only on planar holes. Other solution would be to draw a missing surface and **Join** them together or simply move one edge to another with **MoveEdge** command.
It is a good practice to merge two coplanar faces after fixing is done as it will generate cleaner mesh at the end of the process.



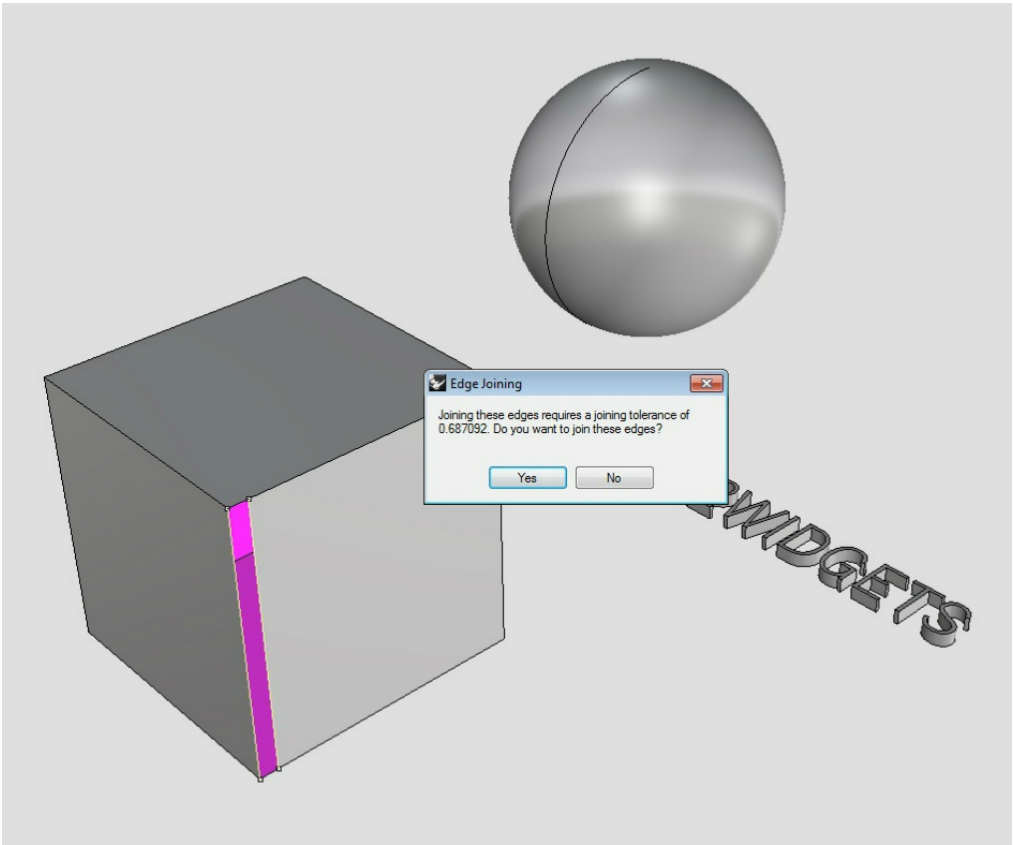
In cases of more complex geometry, it may be useful to create a missing surface from edge curves, surface > edge curves or



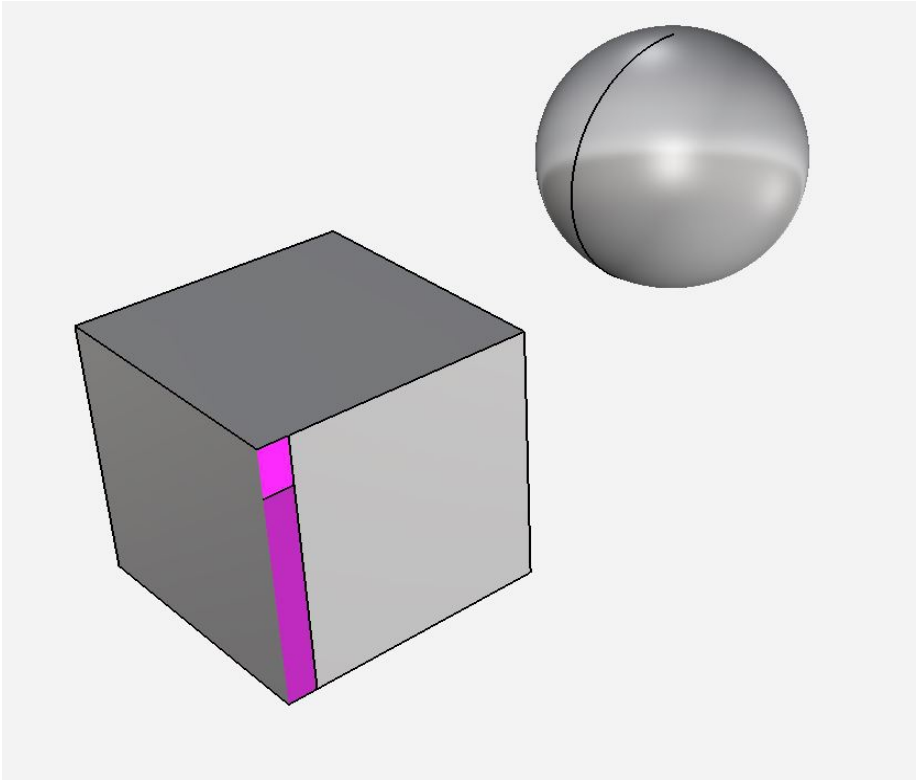
This command works only with 2,3 or 4 edge curves so if the missing surface has 5 or more edges it may be a good idea to use it in two or more stages. This command is very rarely used but in fact produces much better results than the **Patch** command.

If the quick fix is needed and the size of the gap is is not influencing geometry too much there is a command that solves these kinds of issues. **JoinEdge** command forces Rhino to close the gap between two naked edges and although it may not be visible, once the object is meshed the gap will be filled

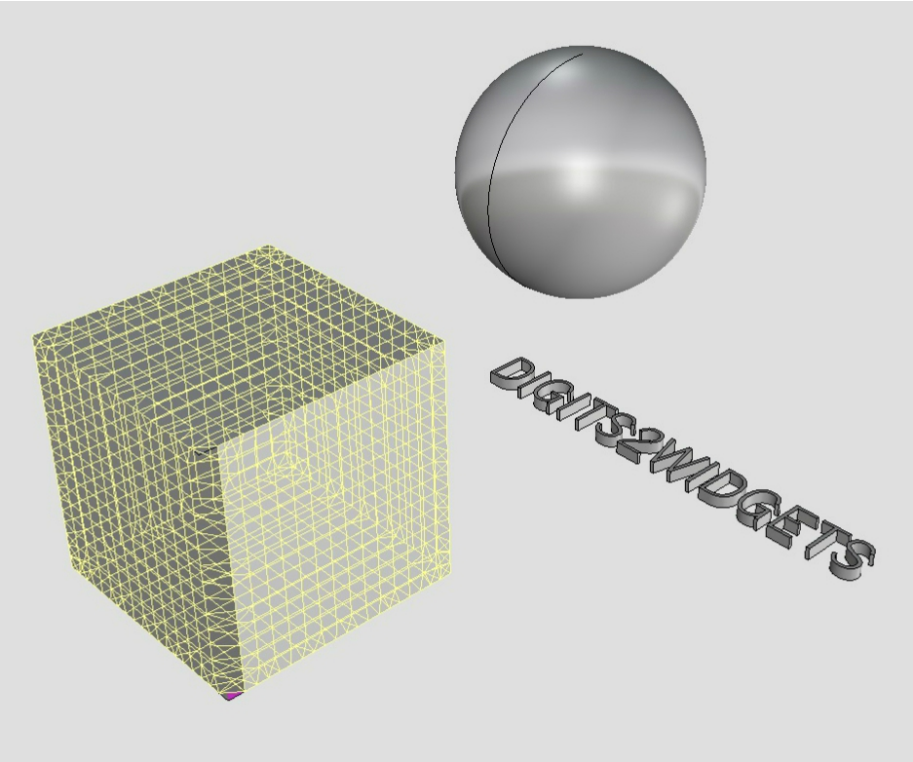
Step 1 – Joining Edges



Step 2 – Joined Edges



Step 3 – Applied Mesh



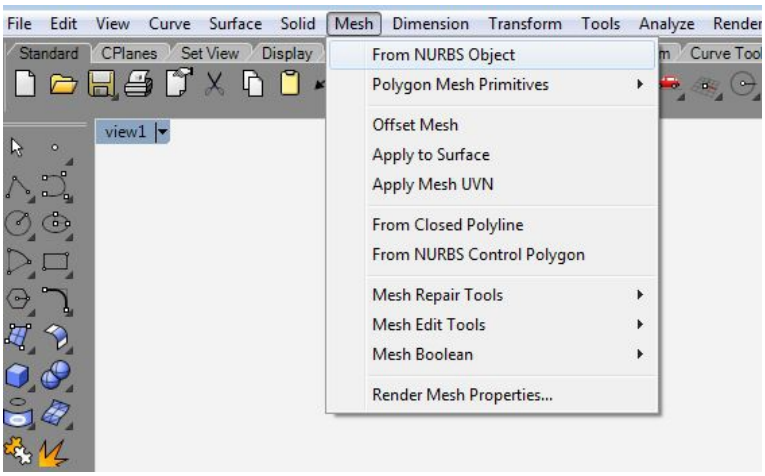
Fixing open meshes can be very tricky and time consuming. McNeel has a very good, but long guide that has been created for Rhino 3 that can be downloaded here: http://www.rhino3d.com/download/rhino/3.0/stlrepair
Most of the tools and commands mentioned in this linked tutorial can be now found in the **Mesh tools**, **Stl tools** or **Analyze** toolbars.

Correct Settings for .stl Export

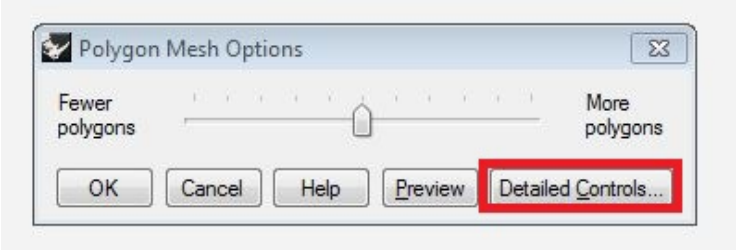
Once the design is complete it is time to export the desired geometry to a .stl file. There are 2 main ways of doing that, but we'll focus on one only.

In order to create a good quality print we need to **Mesh** polysurfaces and surfaces at the suitable resolution. What this means is that a mesh is a network of flat triangles and if the resolution is too low your print will not have smooth round surfaces and will appear faceted. If the resolution is too high the file produced will be too large for the 3D printer to work with or will simply take hours to download.

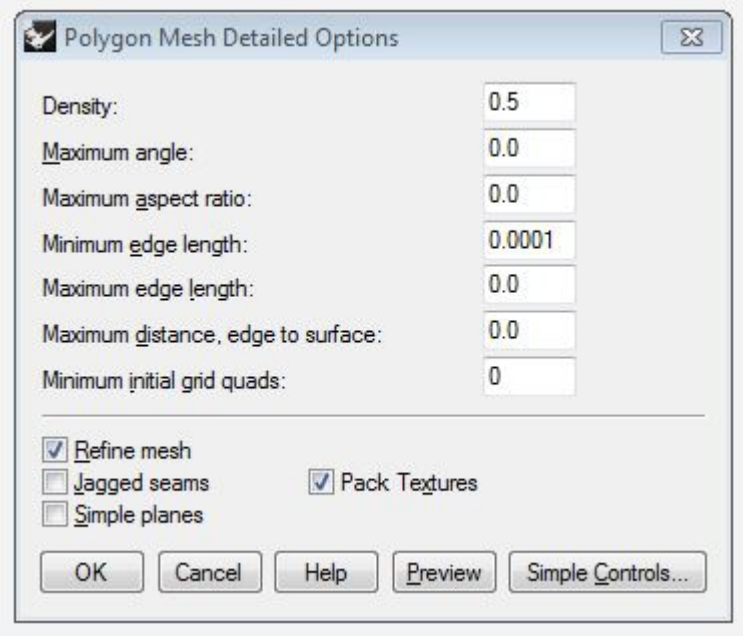
To mesh a polysurface / surface you need to use **Mesh** command



The following box will appear



and we need to get more control over the options by pressing **Detailed Controls** button.
Once enabled it gives us the access to a much more advanced menu



Each of these settings will influence the final output and depending on the level of the detail required in the final .stl file they need to be changed.
A good starting settings would be as follow:

Density	1
Maximum angle	20
Maximum aspect ratio 1 or 2	1 or 2
Minimum edge length	0.01
Maximum edge length	0.0
Maximum distance, edge to surface	0.01
Minimum initial grid	50-200

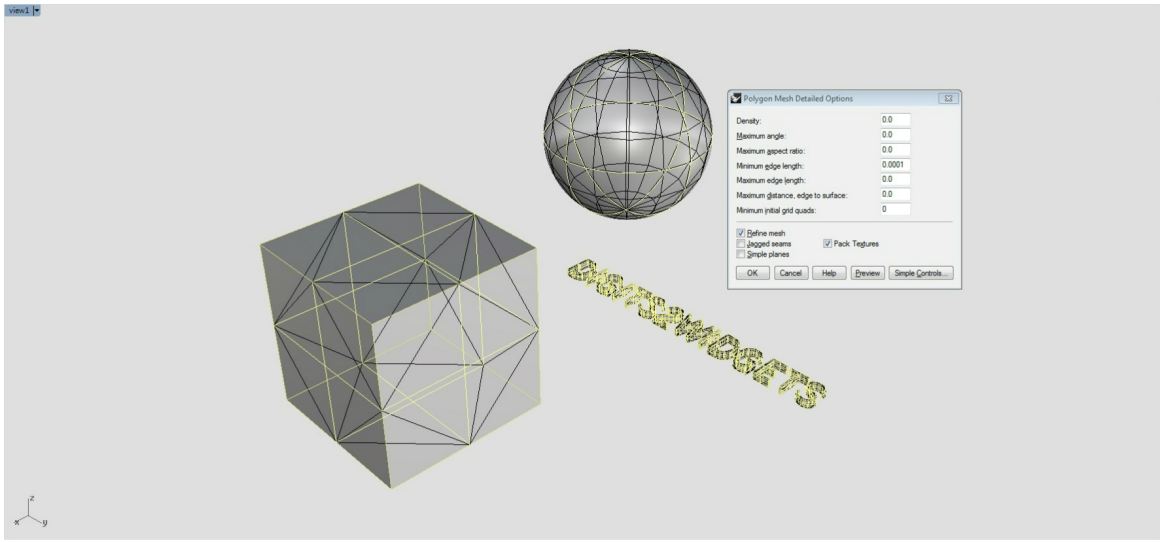
Simple planes checkbox leave unticked

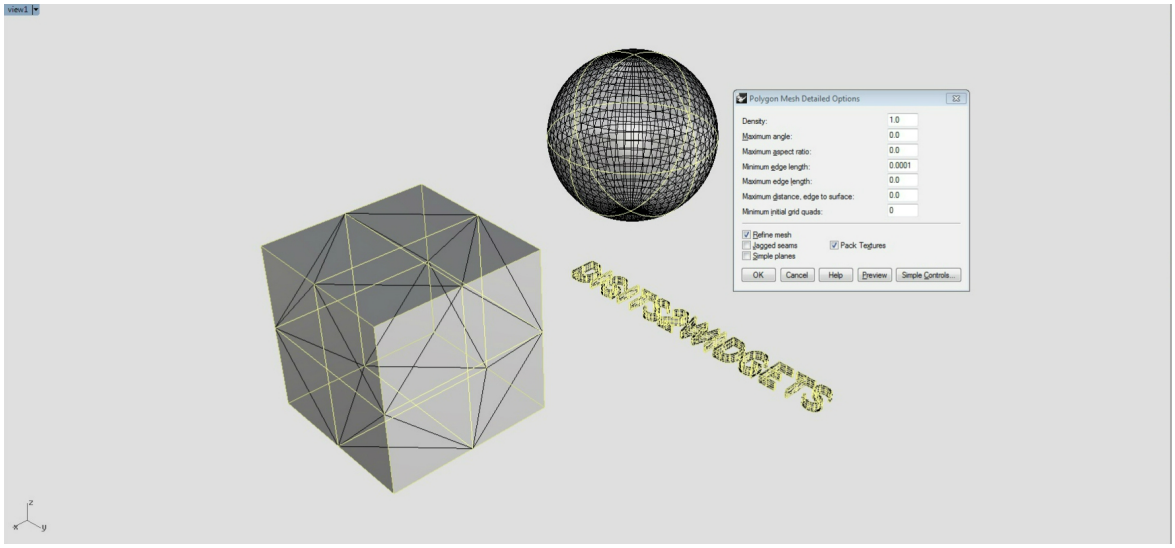
These settings are proven to work in most cases. If the design is extremely detailed and has a lot of organic shapes at a very small size (mainly in jewellery) settings like **Maximum angle** and **Maximum distance, edge to surface** may need to be changed. Lower the angle and decrease distance.

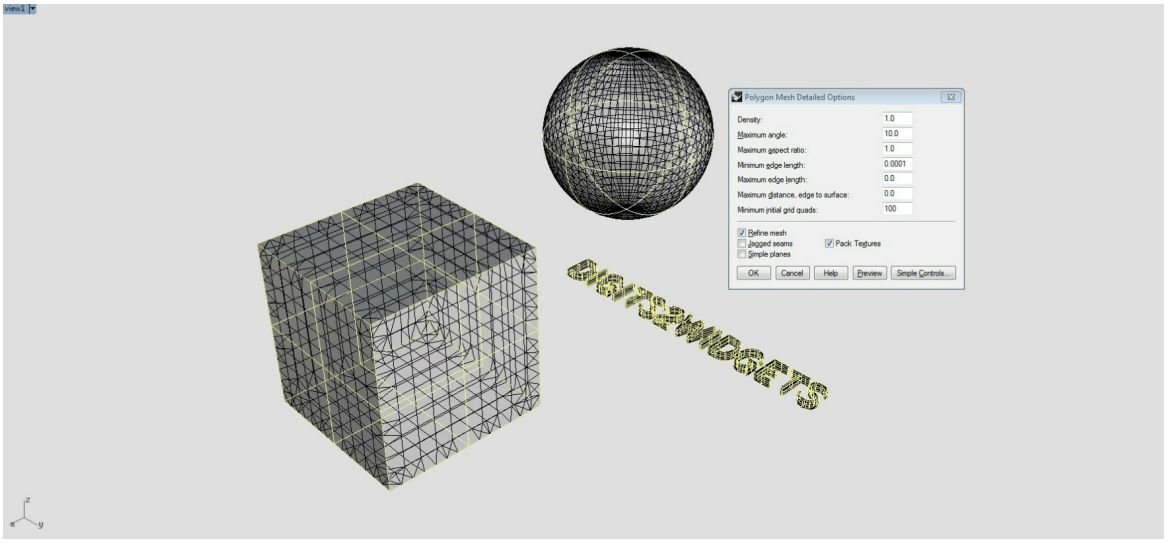
It is always a good idea to use **Preview** button to see the effect of any changes in the mesh settings

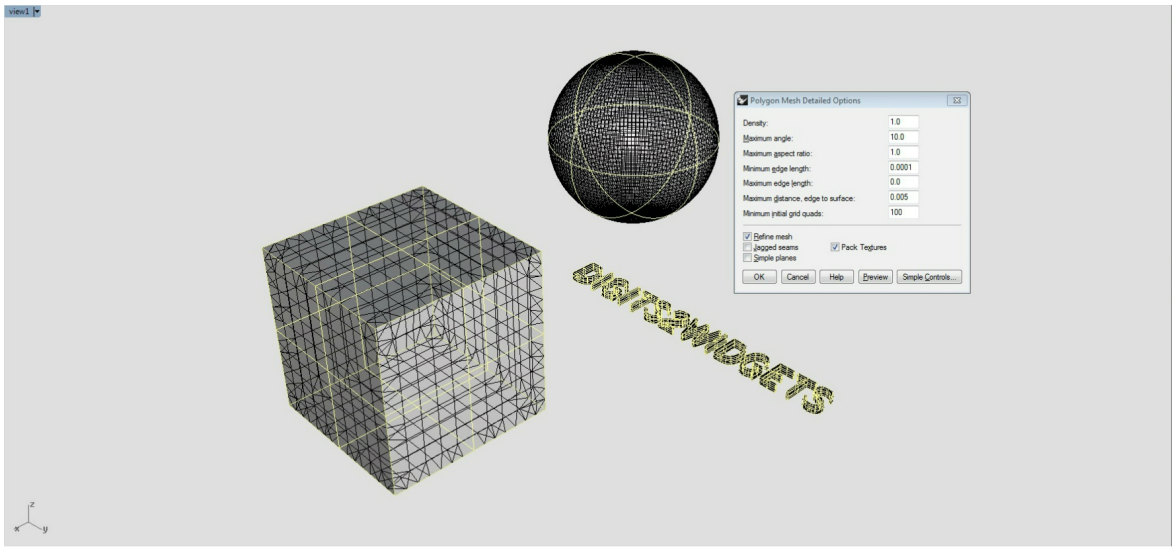
If the file contains lots of or only flat surfaces it may be beneficial to lower **Minimum initial grid** to 10-20

Here are some examples of different meshing settings tested on the same objects:

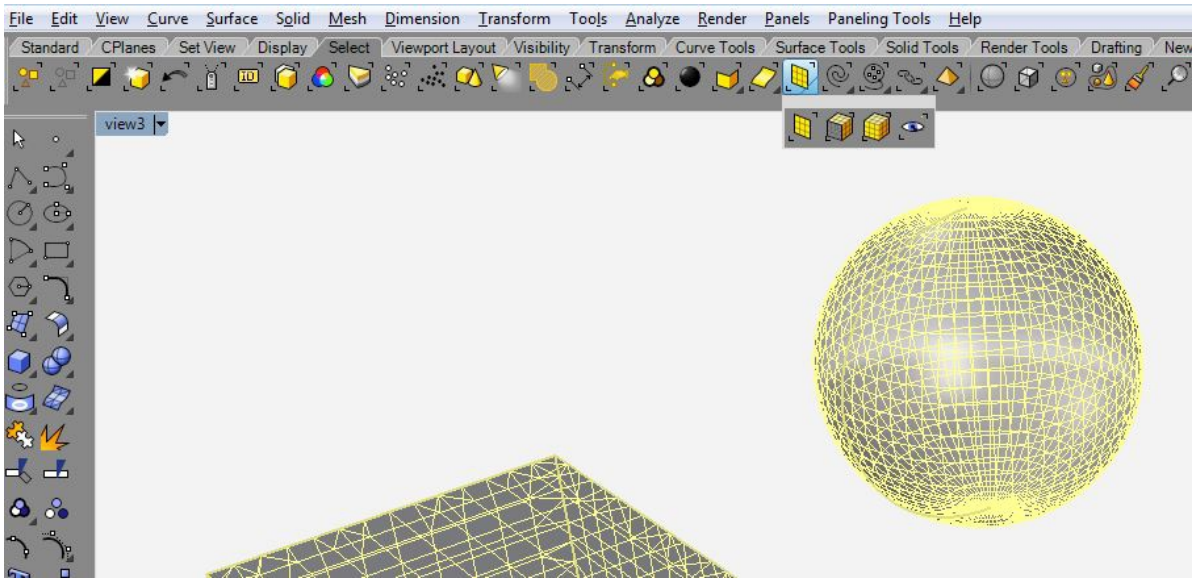








Once you're happy with your settings accept them by pressing the **OK** button. Now you need to select all of the meshes by typing **SelMesh** or by going to the **Select** tab.



With your objects selected type **Export** or go to menu > file > Export selected and choose destination folder and a name for the file. Once exported it is a good idea to compress it (zip) as it will reduce the size of your file dramatically.

If you have any more questions don't hesitate to call us on +44 (0)20 3697 7969