Beginners guide to 3D printing facets from TurboCAD

This document simply shows some of the options available, when creating facets on a 3d object, prior to saving TC drawings for 3D printing. This differs depending on whether one is using Deluxe / Professional (TC surfaces), Or Platinum version (ACIS objects).

It can not go into every scenario, but hopefully gives an overview of the settings.

Version 1.0

Andy H

Important.

This doc may show some ways to increase the number of facets. **However, in many cases this is overkill.** In deluxe, setting approximation lines to 88, or on platinum setting ACIS to 100, will often provide very good results. This is especially true with newer slicers. Therefore I suggest, resisting the temptation to squeeze as many facets out of of TC as possible. As this can slow down TC, and do nothing to improve the look of a printed model. Also some slicers cannot cope with a huge number of faces (triangles).

If an object has many flat faces (angle iron for example). And only a few curved object (one bowed section)). It is simply not worth spending time trying to get it smooth. Simply use the 88 / 100 settings and live with any slight faceting that may occur.

If is an object has lots of curved surfaces, (A golf ball for example). Faceting will simply not show up, due to the amount of changes to the curve, Any faceting that may have occurred on a sphere, would be hidden in the dimples. A setting of 88 (Tc surfaces) / 100 (ACIS) should be fine.

Contents	Page	Contents	Page
Facets	2	Smesh	23
TC surfaces	4	File Formats	25
Symmetry	5	Suggestions	28
Non native objects	7	Calculations	29
Arc/Polyline v Spline/Bezier	7		
Cylinder alternative (Bezier)	8		
Join polyline	9		
ACIS objects	10		
Expert Faceter	11		
Terms	12		
Surface tolerance	12		
Normal tolerance	15		
Edge length	16		
Grid lines	18		
Spline / Bezier	21		

Facets (Faces).

Much of this document will be talking about facets or faces. In 3D terms, a facet is a flat face on the surface of the object. Sometimes these are triangular, and sometimes rectangular.

TurboCAD allows for a different amount of facets depending on the user setting, and whether using Deluxe / Pro. Or Platinum.

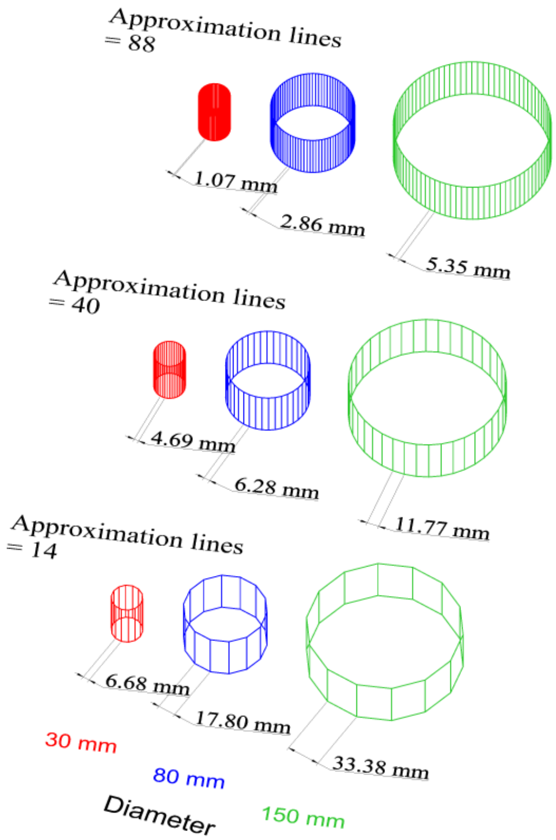
Fig 1. Shows the facets applied to TC surface object, The edge length (facet width in this case), demotes what may be printed as 'flats' by the printer. However, some slicers / printers may automatically round over some facets, making the object smoother.

How many facets would be fine depends on the slicer / Printer. Inside TC the amount of facets affects how good the render will appear. And the more facets the better the render. This is not always the case with 3d printers.

Looking at the diagram, one would not want to print at 14 approximation lines, That is evident even from the render / wire frame.

However because of lighting etc. and the fact one can zoom in on the screen. One cannot always say a print will be smooth, or faceted, just from within TC.

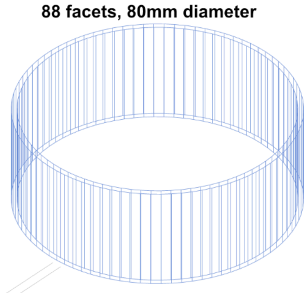
Fig 1



Looking at Fig 2. It looks quite faceted within TC. But from Fig 1. We see it is only 2.86 mm facet width. This may sound a lot, but some slicers, will jut see this as acceptable for a curved surface. And any faceting will be hardly noticeable.

Fig 2

88 facets, 80mm diameter



Just to show in real life. Fig's 3 and 4, show the printed objects at 14 approximation lines and at 88

Fig 3

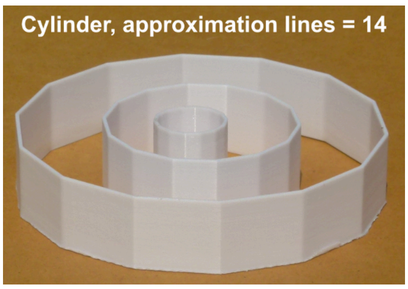
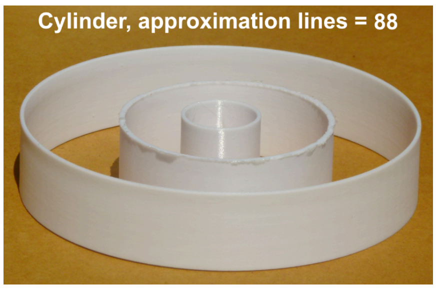
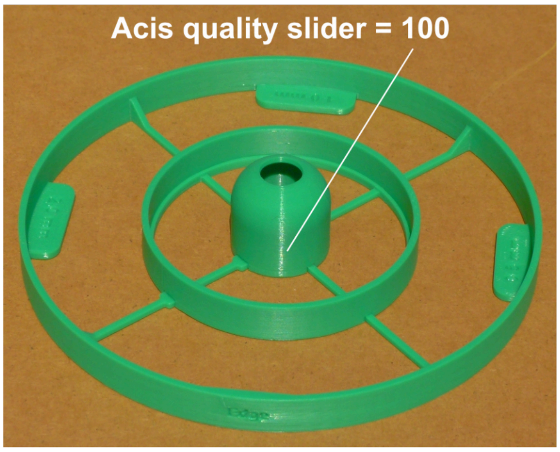


Fig 4



To check the facet quality at varying edge lengths. I designed these two little objects, where each quarter has a different facet edge setting.

Fig 5

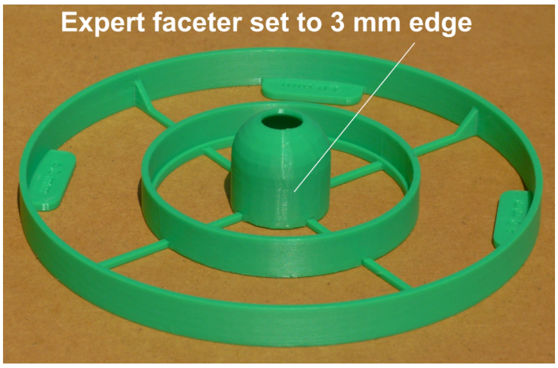


The first, Fig 5 has ACIS 100, Edge 0.5 mm, 1.0 mm and 2.0 mm.

The second, Fig 6 has 3 mm, 4,mm 5 mm and 6mm

This shows that facets are far more pronounced on smaller objects. This is true for any facet using edge length.

Fig 6



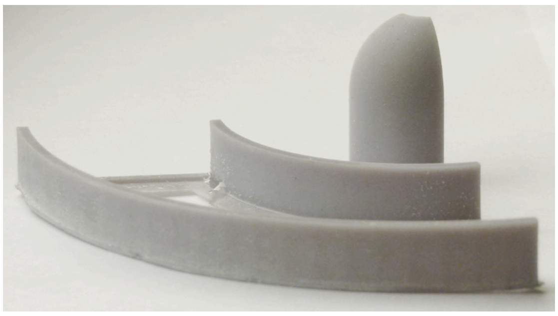
Whereas Normal tolerance and surface tolerance, is the opposite, with those, the larger the object, the more chance of facets occurring.

It is difficult to see in the images, but with the quality slider set to 100, one can just make out some faceting on the larger arcs. Whereas the small object is smooth. This is because the ACIS slider uses normal tolerance to set the facets.

With Edge length, ever at 4 mm, the outer arcs are quite smooth, but the small inner object gets progressively worse.

That said, shallow curves are less likely so show any great faceting, because the molten plastic will just blend itself at corners, rather that come to a stop and then continue down the next facet.

Fig 7



This document is primarily aimed at FDM (extruded plastic) printing. Resin printers may show facets differently to FDM because of the increased printer resolution.

If using a resin printer. It may be necessary to increase the faceter quality.

Fig 7 shown a more faceted appearance on the larger arcs. As the resins higher resolution (lower figure), means the corners are not blended as much as FDM.

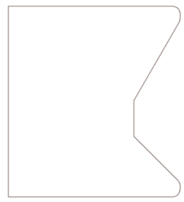
The inner dome looks good, as the model was printed from using ACIS slider set to 100. Which uses Normal tolerance, thus the edge length of the facets are closer together, the smaller the arc is.

TC Surfaces.

TC surfaces does not have subdivision. By that it is meant that once one has set the max facets for a native object, one cannot divide the facets to produce a smoother looking surface. In some other software one can divide facets to produce a smoother (less faceted) curve surface, when the object is 3d printed.

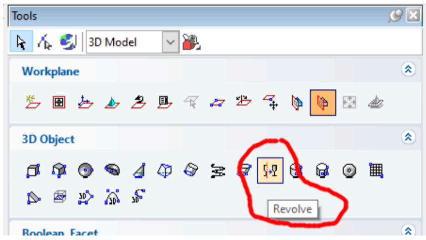
NOTE. In the following, we discuss changing the segments. This only applies to created object, Cylinder, Revolve etc. as shown at the top of the selection palette. If a Boolean or any other operation is carried out which changes the description to a TC surface, the ability to change the number of segments is lost. Therefore, always set the correct number of segments, when the object is created, and certainly before carrying out any Boolean operations, or exploding the object.

Fig 8



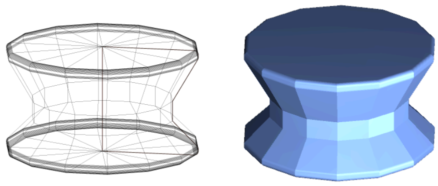
For the first test, we will use a revolve. We start with the 2d profile as in Fig 8.

Fig 9



Select the revolve tool, shown in the tools palette in Fig 9. Or the menu (location may vary depending on the user interface). Or the 3D toolbar

Fig 10

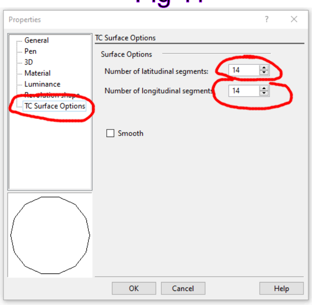


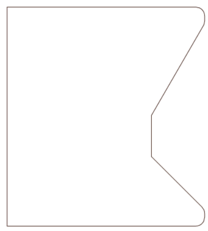
Looking at the finished revolve in Fig 10. We can see it is not round. TC surfaces are faceted by nature, and as such does not look round, i.e. they produce many flat faces to represent a curved object.

In Fig 10, the number of approximation lines are left at the default 14 segments around the object.

We can improve the look by selecting the object, then, right click the mouse and choosing properties from the menu. Or click the properties icon, bottom left of TC program. Or using the properties section of the selection palette.

Fig 11





In the properties dialog (Fig 11), Click on TC surface options and set the number of segments to a higher number. I suggest using 88. The max allowed is 90.

Higher numbers can slow down TC, if many object are curved and all have a high segment setting. Though modern computers should be more than capable of coping with most situation.

Fig 12

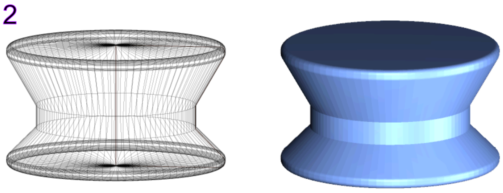


Fig 12. Shows the result after we have change the segments to 88. Whilst it is still faceted in appearance, the extra faces make the object look much smoother.

Fig 13

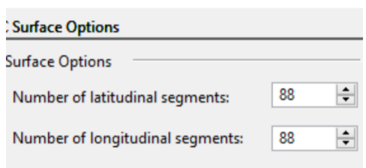
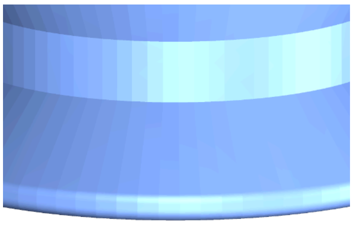


Fig 14



Zooming in to the object, Fig 14. Shows the object is still faceted, but looks a lot closer to a circle than the original, which only used the default 14 segments. And the object should 3D print without leaving any noticeable facets, certainly on smaller to medium objects.

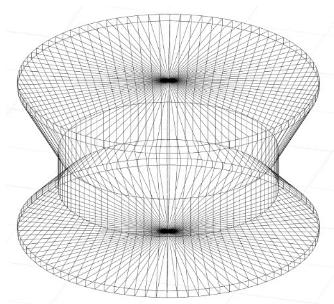
A brief explanation of the segment settings.
Say we have a cylinder drawn as 40mm diameter, if we use 88, the inspection bar will still report both X and Y as equal sizes. This is because the bounding box of north, south, east and west, all lay against either all flat faces or all vertex points. This relates to any cylinder, revolve etc. where the total segments can be divided by 4. If we used the maximum of 90, the inspection bar will read 40 and 39.9756. This is due to the bounding box, laying against two flat faces, and two vertex points.

A quick note about the 'smooth' option in the objects properties dialog, shown on the previous page. This option is cosmetic. In that is doesn't physically alter the object, but makes it look smoother when rendered. Leave it turned off when setting details for 3D printing.

The previous page stated '' selecting the object, then, right click the mouse and choosing properties from the menu.''   We can negate this by setting the properties for the tool, before any object is drawn.   i.e. select the tool, the right lick and choose properties. Set the desired properties and click OK. Future objects drawn within the drawing, will use these new settings.

Say we would like it even smoother, or closer to an actual curve object.

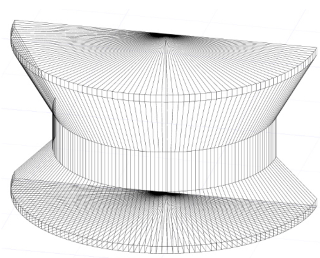
Fig 15



The main thing to look for is symmetry.

Looking at our origin 360 degree rotation, set to use 88 Segments, Fig 15.   The object is in perfect symmetry. The 88 segments go all around the object. If we explode the object once, the selection palette reports 616 faces. This figure includes faces on the top and bottom.

Fig 16

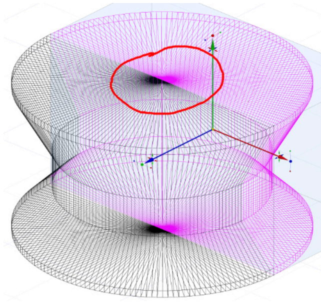


If, instead of 360 degree rotation we revolve 180 degrees, and subsequently mirror copy and Boolean add the two object. We can obtain the full 360 with more segments. As the '88. Would originally be drawn for 180 degrees, the 360 would have 176 segments.

Comparing the object in Fig 16. And comparing it with Fig 15. We can see that the result is more facets, that can 3d print smoother. After mirror copying and Boolean adding, the selection palette reports 872 faces.

We can go even further and revolve 90 degrees to produce a quarter of a circle. And create the other three quarters for the full 360 degrees. **However,** the more times this is done, the greater the chance of a failure occurring. Plus the slower TC could become. There also comes a point where more segments / faces makes no difference to the finished 3d print.

Fig 17



Occasionally, the previous method of using a mirrored 180 degree revolve and Boolean adding, may result in the Boolean add failing.

This appears to be due to where all the top and bottom triangle meet in the middle. If the meeting isn't correct on the mirrored copy, a Boolean operation could fail. Fig 17. Shows the meeting point.

Fortunately if this problem is encountered, there is a simple solution. First we need to slightly modify the original profile, there are many ways to do this, but here is one method.

1). Draw a rectangle at the left hand edge of the profile Fig 18.
2). In edit mode, grab the two nodes at the left of the profile, and intersect snap them to the right hand side of the rectangle. Fig 19.
3). Revolve the profile using the left hand side of the rectangle as the revolve axis. Fig 20.
4). Carefully mirror copy the object, and Boolean add together. In tests, as all the facets do not merge at he same point, it appears to make Boolean adding simpler.

Fig 18

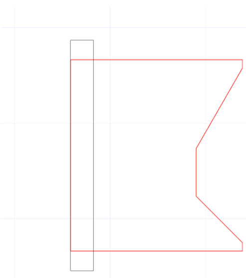


Fig 19

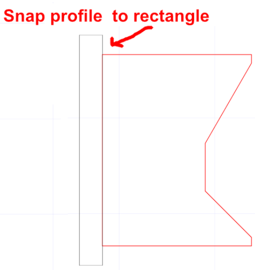


Fig 20

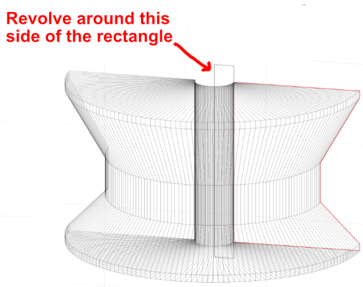
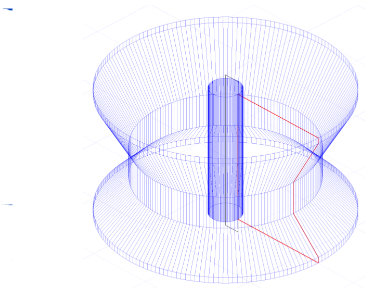
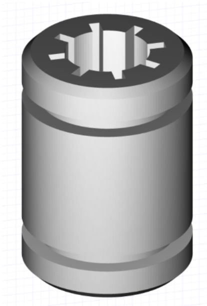


Fig 21



OK we now have an unwanted hole Fig 21. A few methods of dealing with this. Two methods are.

Fig 22



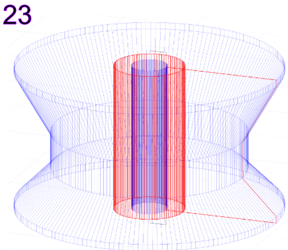
Method 1.

Ignore it. There are times when object will have a hole produced in it, where the above created hole is located, drylin Bearings for example shown in Fig 22.

Therefore there is no need to do anything with the created hole.

Method 2.

Fig 23



Fill in the hole.

Draw a cylinder or box covering the hole, and at the same height as the object, as shown in Fig 23.

Boolean add the cylinder / box to the object, resulting in in a filled and completed object. Fig 24. With 174 facets around the circumference. Resulting in a smoother printed object.

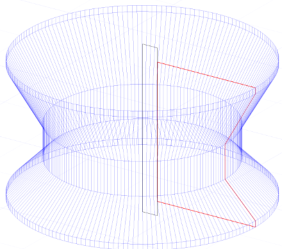
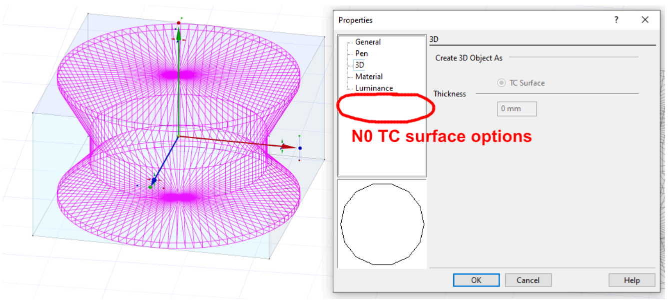


Fig 24

What if the objects is not in its native state.

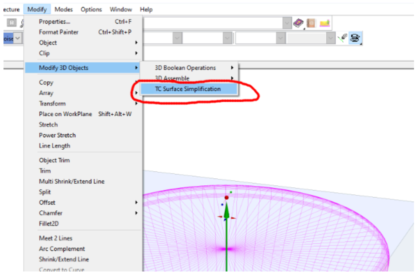
Fig 25



If any Boolean operation or an explosion is carried out on a native object, the ability to set the number of approximation lines (TC surface options) is lost to the user.

There is little the user can do inside TurboCAD, to increase the amount of faces on these objects.

Fig 26



One operation that may have been considered, is using the 'TC surface simplification' tool.

Normally this tool is to reduce the facets on a surface object. But setting it at 100 percent to keep, and clicking finish, will divide the faces, However, it converts rectangular faces to triangular, keeping the faces flat.

Therefore it does little to make the object print smoother.

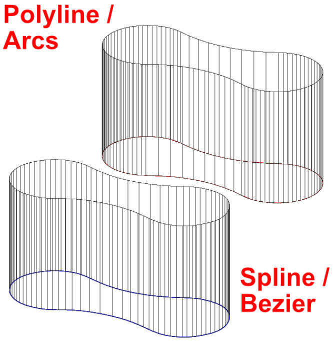
If a print is not acceptable, then this means using a third party application to attempt to smooth out the object. However, this in itself can be problematic, as re-meshing could change the size or the shape of an object.

A good option if TC cannot produce a smooth enough object, may be to use the TC exported object, as a template for creating a new object in a third party application.

An option within TurboCAD is to ensure the number of segments. of all native curved objects including sweep and revolve, is set high (88) prior to carrying out any further operations on the object.

Arcs / Polyline v Spline / Bezier.

Fig 27

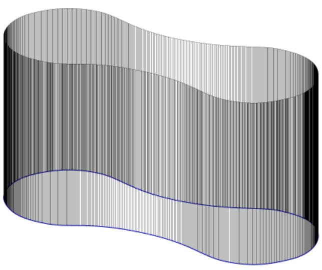


When extruding or revolving standard polyline / arcs and spline or bezier, the extrusion look similar. And is fine for many applications.

Fig 27. Shows both extruded with 88 approximation lines (facets). And as one can see, the results are quite faceted, on the shallow part of the curves. Unfortunately even drawing / extruding a quarter, and mirror copying to get the rest, shows the same problem.

One way to increase the facets, is to use a closed spline / bezier, and explode once down to a polyline. When a spline / bezier is exploded it often creates a lot of vertices trying to match the curve.

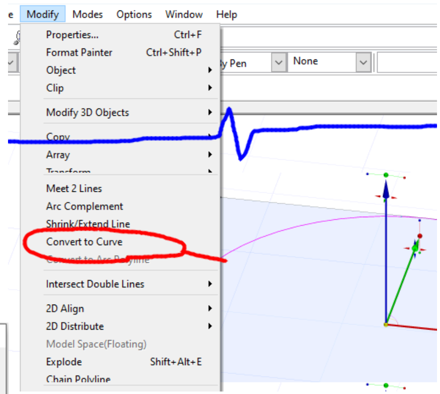
Fig 28



As can be seen in Fig 28. This increase in vertices will produce a smoother looking 3D print, that will likely not show any faceting.

Cylinder alternative.

Fig 29



Continuing with spline / bezier theme. There is a tool called Convert to curves. Location may vary, but is likely to be either format menu, or modify menu.

This tool will change a polyline / arc / circle, to a bezier curve. Based on a selected tolerance. This example uses a circle, to convert to curves.

Select the circle, then select convert to curves from the appropriate menu.

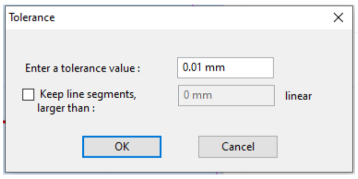
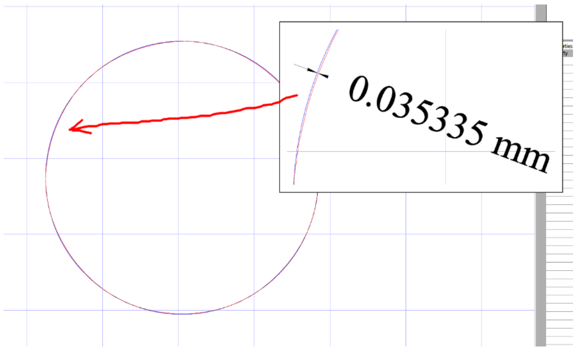


Fig 30

Fig 31

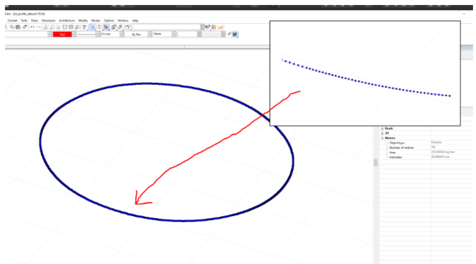


In the dialog that appears, type in the required tolerance. In this instance 0.01mm has been used. Click OK.

As can be seen, (circle = blue, bezier = red), the resultant bezier may not be 100% accurate. And in some cases may require manual alteration.

In this case it is deemed close enough at 0.035mm difference

Fig 32



Next step is to explode the newly created bezier to a polyline. This may seem odd, especially if we started out with polyline/arcs, But exploding the bezier converts it to lots of close polyline segments.

In Fig 32. The new polyline was selected using edit mode. And shows there are a very large amount of nodes (segments). A total of 701 vertices.

Fig 33

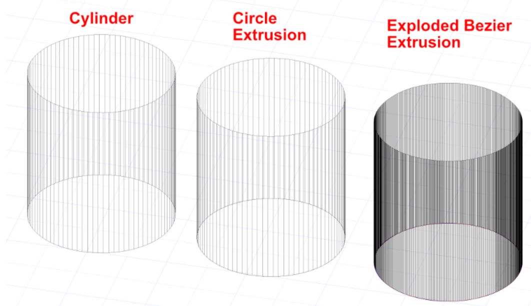


Fig 33. Shows that a cylinder and an extruded circle, are almost identical when both are set to 88 number of approximation lines.

However the extruded bezier explosion, shows many more faces, which would result in a smoother cylinder when printed. Experimentation may be required on the bezier to find the best acceptable tolerance.

Note. Too many objects with a high node count, will likely slows down TC. How much would depend on ones computer spec.

This will not work every time. On a 130 mm diam cylinder, after carrying out the above, We ended with 8000 nodes. Far too many to be useful in TC, and useless for 3D printing.

Join polyline (on a spline / bezier).

Under normal circumstances, one would not use join polyline on a spline / bezier. This is because the join polyline tool, converts spline / bezier into a polyline when creating the join. And can lead to a huge number of nodes and thus tiny facets. Which are not good when using a 2D drawing.

Whilst the use of join polyline will not affect the resolution of a revolve which would remain at 88 if that is the setting in revolve properties. It can make a difference in other aspects of a revolve.

In Fig 34. Is a polyline with arc, revolved 360 degrees. Fig 35. Uses a polyline and a spline instead of an arc. The join polyline tool is used to add everything together, which changes the spline to a multi-node polyline.

Fig 34

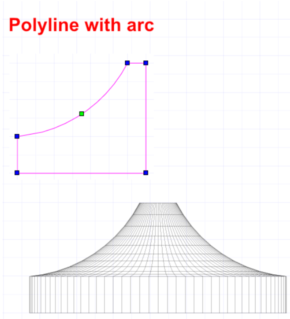
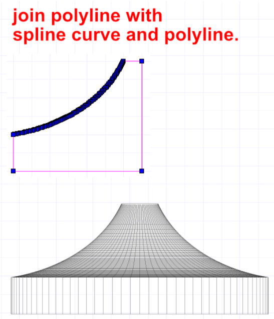


Fig 35



The resultant revolve means more facets in the V direction (up). Though as previously stated, it cannot alter the facets in the U (around the object). Which remain as set in the number of approximation lines.

For the approximation lines one would still need an alternative approach, like revolving 180 degrees, or using another software.

Fig 36

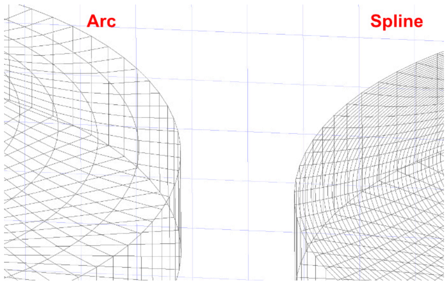
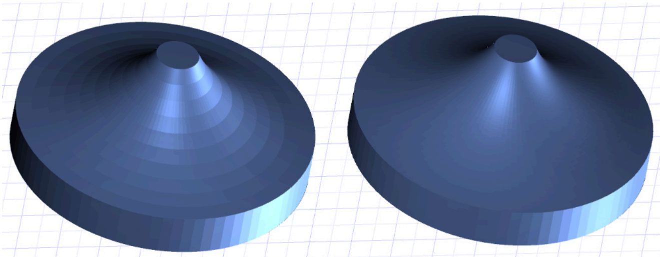


Fig 37. Shows the rendering, and how much smoother the curve looks on the right, compared to the arc curve on the left.

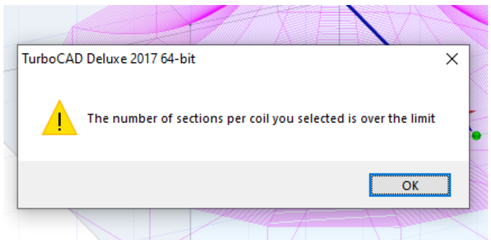
Fig 37



One drawback, you can go too far with the nodes. If this happens, TC will pop up a warning message, Fig 38 saying you have selected too many sections.

You need to reduce the number, or click OK, and TC appears to automatically reduce the number to the maximum allowed.

Fig 38



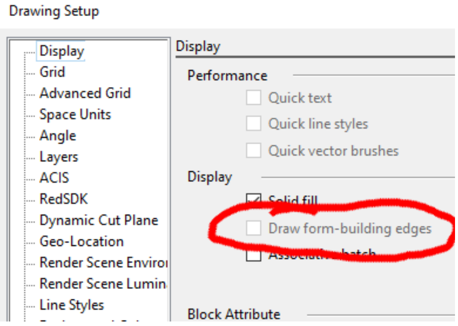
This can unfortunately, make the part worse than than if it was left as an arc. It is simply a matter of experimenting, as to what works best in a given situation.

ACIS objects,

TurboCAD platinum allows for the use of ACIS solid modelling objects, and this allows for more control over the number of facets associated with an object.

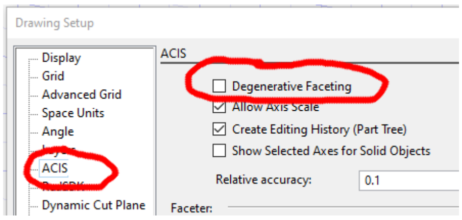
To see this control before saving for printing, ensure draw form building objects is turned on, located in options menu - drawing setup. Fig 39.

Fig 39



NOTE, if using RedSDK, this option is greyed out, because the setting is turned on automatically and cannot be altered by the user.

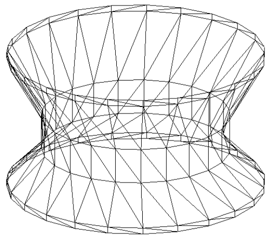
Fig 40



Secondly, ensure Degenerative faceting is turned off. Located in options menu (drawing setup) - ACIS. Fig 40.

Fig 41

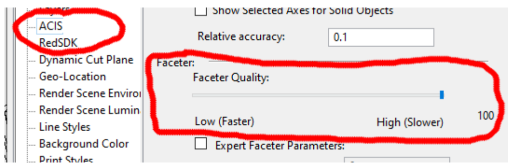
Fig 42



Using the 2d profile as for TC surfaces. Fig 41. We revolve the profile as an ACIS solid. The result at default ACIS settings is not encouraging for 3d printing. Fig 42.

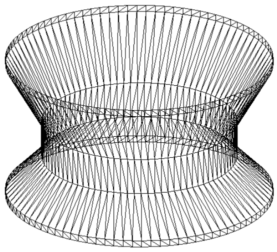
This is because the default ACIS Faceter quality setting is just 50 percent.

Fig 43



To improve the faceting, we need to increase the faceter quality to a higher setting. In Fig 43. The setting has been changed by dragging the slider up to 100.

Fig 44



The result is shown in Fig 44. And shows that 100 percent ACIS is not much different to the 88 segment setting of TC surfaces.

The difference with ACIS is that we have the ability to improve the segments.

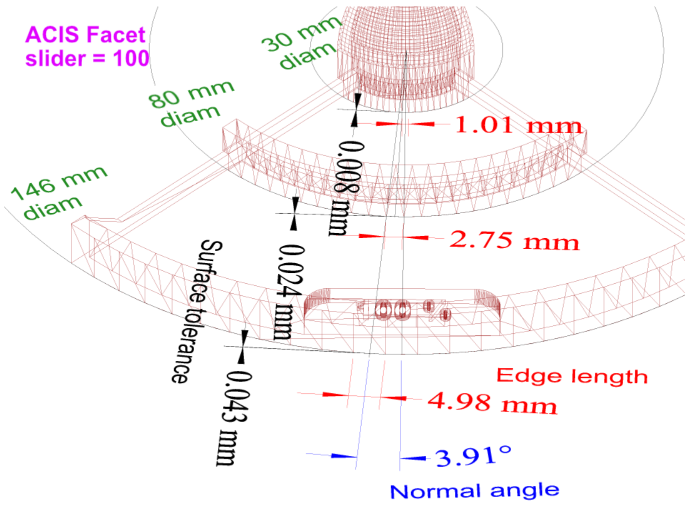
NOTE. In some versions of TC, changing the faceter setting, will <u>not update</u> the drawing after clicking the OK button in the dialog box. This appears to be linked to having degenerative faceting turned off.

There are a number of ways to get round this. For example.

1). Wait until ready to save the file for 3D printing, before adjusting any settings.
2). In the ACIS (drawing setup) dialog box. Adjust a setting and turn on degenerative faceting, click OK. Re-open the dialog, and turn Degenerative back off.
3). In the drawing select the part. Open the selection palette. Against 3D change the setting to Surface. TC should pop up a warning dialog (assuming the user has not turned warnings off). In the warning dialog, click cancel. This will update the part whilst leaving it as an ACIS solid. Use this method with caution. In case the popup does not appear.

4). Operate with degenerative turned on, and an ACIS setting of 90 or 100. When one wishes to view what the wire frame for 3d printing would look like. Make a copy of the part, and explode the copy once. Then delete the copy afterwards. This can be quicker then changing settings in a dialog box.

Fig 45



Before going into the Expert faceter settings, we will take a quick look at the comparison between the slider and the expert faceter settings.

The slider appears to use the facet normal to determine the amount of facets. In Fig 45. We can see that the facet (edge) length, and the surface tolerance changes depending on the diameter of the arc / curve being used.

This means, that the smaller the object, the better chance of the printer to produce smoother 3D prints.

Expert faceter.

In many cases, the setting of 100 in the ACIS facets dialog, is perfectly fine, and will print well. However there are occasions, especially on large parts where more facets may be needed. This is where the 'Expert Faceter Parameters' is used.

There are two ways to change the Expert faceter.
The Options menu - drawing setup - ACIS dialog, sets the global ACIS settings for all 3D solid objects.
OR by selecting an object, going into the objects properties dialog, one can adjust the facet setting for that one object.

Global settings. Fig 46.

Opening up the options menu - drawing setup dialog, and going to the ACIS page. We have a setting for Expert faceter. Clicking this reveals various settings. (i.e. they are not greyed out).

Object ACIS properties. Fig 47.

Help file says *''Faceting can be controlled on an individual object basis via the Properties of the object. Object specified faceting overrides the global setting in the ACIS settings. ''*

NOTE. On my computer, Object properties ACIS faceter setting, does not work for most TC versions. In v2021, it works, in specific circumstances. Making it more trouble that it is worth for me. Your findings may be better. This may be due to my TC configurations.

Fig 46

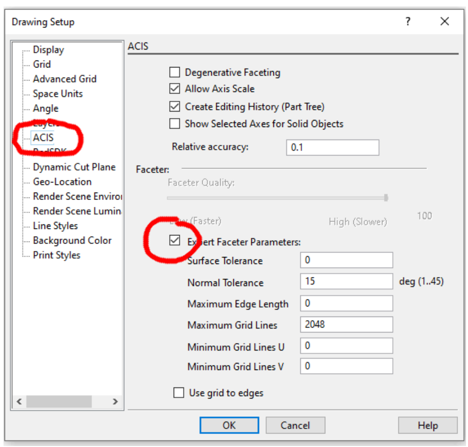
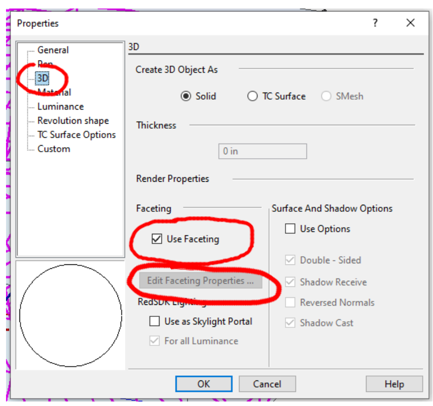
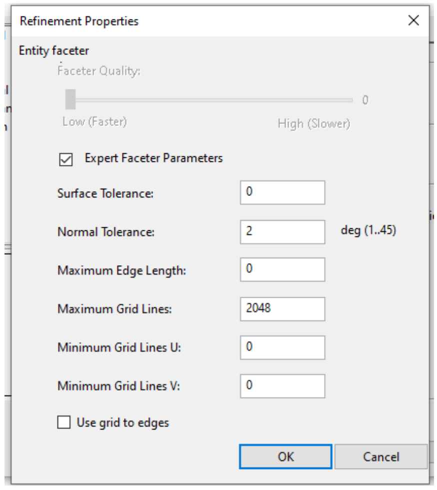


Fig 47



Also, as with global settings, the expert faceting may not update straight away with degenerative faceting turned off. (see bottom of 10 page).

Fig 48



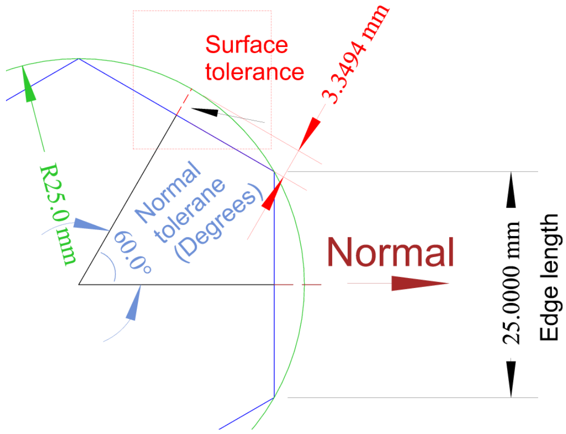
For both global and object faceter settings. The expert faceter brings up a number of options Fig 48.

These settings are interlinked, therefore there is no need to adjust all the settings, Example - Surface tolerance and normal tolerance have a similar end result as far is the amount of facets displayed, But do it in different ways.

Expert faceter terms. Fig 49.

Surface tolerance. This is the chord height between a facet edge and the true dimension. The true dimension could be based on a circle, arc, spline or bezier.

Fig 49



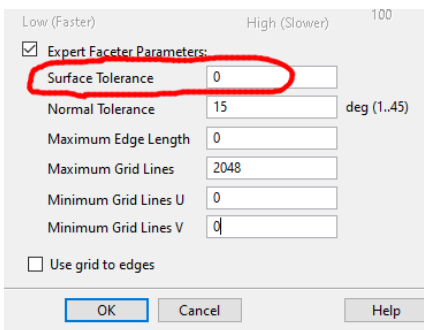
Normal Tolerance. This is the angle between two normal’s, A normal is a point perpendicular to a face. Generally, for ease of visualising, I class the normal as emanating from the centre of a facet.

Edge length. This is the easiest to visualise, and is the length on a single facet.

For ease, the following examples, use a revolve, or cylinder. This was done so as to attempt to keep all the facets the same size. Spline and Bezier, will often have facets of varying size, resulting in a wide variation in edge length for different parts of the curve.

Surface tolerance. Fig 50.

Fig 50



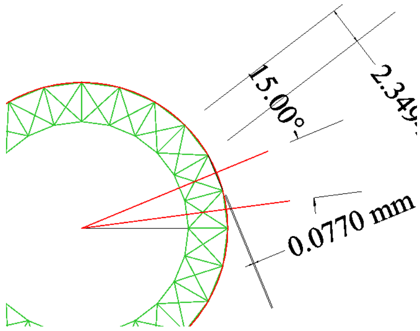
IMPORTANT NOTE. Surface tolerance is not working in my V2017 and v2018. Unknown if it is my system, or a problem with TC, the tests for surface tolerance were done using v2019.

From the help file “ The Surface Tolerance represents the maximum distance between a facet and the true surface definition. By default this value is a predefined fraction of the body diagonal. However, using the Expert Faceter Parameters, you can override the default behaviour and precisely control the deviation between the facet and true surface. For example, by specifying a value of 0.05, facets will deviate no further than 0.05 inches/mm from the true surface. This is particularly useful to remove facet display anomalies due to tolerances. “

Basically. Decreasing the tolerance, increases the amount of facets used to represent the object.

Turning on expert faceter parameters, without altering surface tolerance (leaving it at ‘0’), Effectively disable the function (Note I am not a programmer, this assumption is based on experiments). Instead the program uses the other settings, in this case the normal tolerance.

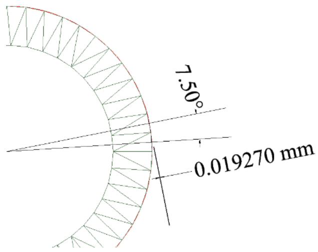
Fig 51



As the default for normal tolerance is 15 degrees, Fig 51. The resultant default faceting is quite course and may not 3d print well, (too faceted) on large objects.

To use surface tolerance we need to change the figure fractionally above ‘0’. If we look at the default surface tolerance set by the normal tolerance of 15, we can see it gives a surface of 0.077 rounded. Therefore anything above this figure will be ignored, in favour of the normal angle.

Fig 52



Example. (Using 18 mm diameter object)

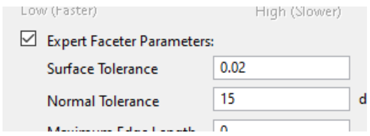
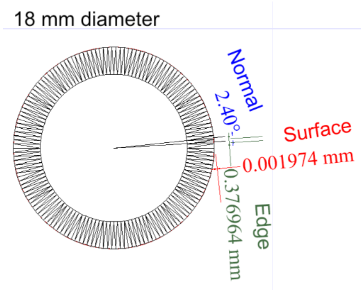


Fig 52. Was set to a surface tolerance of 0.02. This equates to a normal tolerance of 7.5 and an edge length of approx 0.628 mm
Meaning 48 facets around the 18 mm diam object.

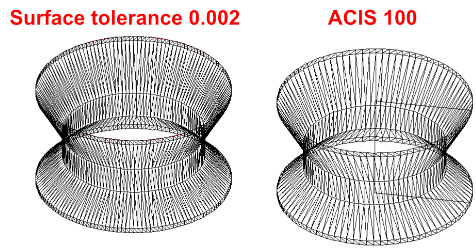
Fig 53



A setting of 0.002 equates to a normal tolerance of 2.4 degrees, and an edge of .376 mm Fig 53. Giving 150 facets around the cylinder for a very smooth finish.

To put that into context, Using normal ACIS setting (without expert faceting) of 100 is approximately 0.0056 surface tolerance, edge length approximately 0.64 mm, and 4 degrees normal tolerance. Giving 90 facets.

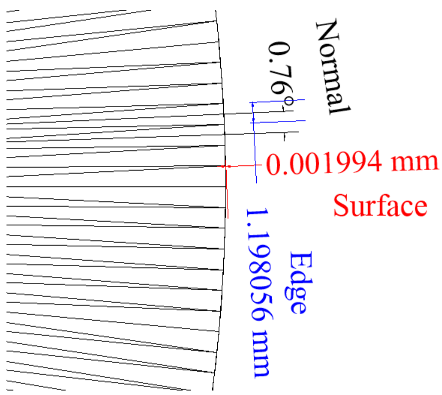
Fig 54



A comparison of surface tolerance, versus ACIS standard setting of 100, is shown in Fig 54. As can be seen, decreasing surface tolerance produces more faces and thus should print looking smoother.

Calculating the balance between surface tolerance, object size, and any affects on TC performance, can be trial and error.

Fig 55



The above objects were 18 mm diameter. However using the figure 0.002, on a larger object, say 180 mm diam, will change the normal angle, and the edge length.

The normal tolerance, surface tolerance and edge length of an object 180 mm diameter, are shown in Fig 55 .

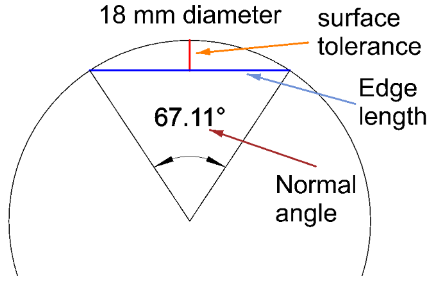
The Normal angle is actually below the minimum one can manually set as the normal tolerance, which has a minimum of 1.

The drawback, if one has ‘draw form building edges’ (options menu - drawing setup) turned on (or greyed out), and degenerative faceting (Options - ACIS) turned off, which will show all the facets of an object. TC can become messy, difficult to select objects, and slower to operate when selecting such objects, with a really large facet count..

One may wonder why the normal angle has decreased so much, between 18mm and 180 mm.

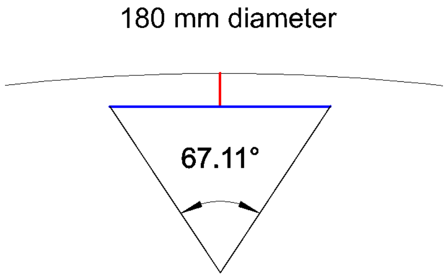
When dealing with a perfect circle (cylinder), all the edge lengths are equal. thus, the angle from normal to normal is equal to the angle between the edge length vertices, to the centre of the circle. This makes it easier to visualise what is going on. NOTE, This applied to cylinders, spheres etc. Also note, the following is exaggerated for clarity.

Fig 56



Looking at figure 56. We can see the surface tolerance shown in red, is between the facet edge, and the true circle. The edge is the chord in the circle. The normal is shown as the vertex angle for clarity,

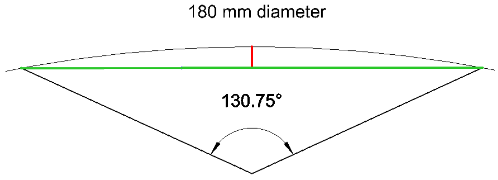
Fig 57



If we then take the same surface tolerance up to a 180 mm circle. We can see the edge length is too short. Fig 57.

Therefore we need to extend the edge length to meet the circle. Which will change the angle.

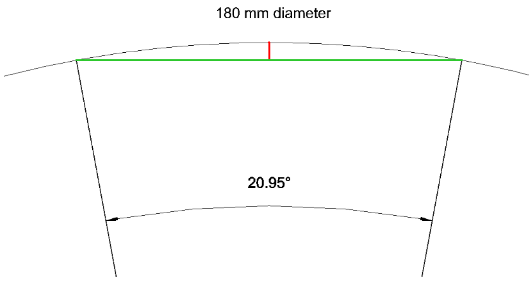
Fig 58



The new edge length has been created shown in green. Fig 58.

The angle extended to match the new edge vertices. Although the angle has initially increased, it should be drawn to the centre of the circle. Therefore we need to move the angle centre vertex, to the centre of the circle.

Fig 59



In Fig 59. After the Normal angle has been reset to the centre. The angle has decreased from the initial angle of 67.11, down to just 20.95

With surface tolerance, whilst the edge length can increase, the normal angle can decrease.

Surface comparison for different diameters.

Fig 60

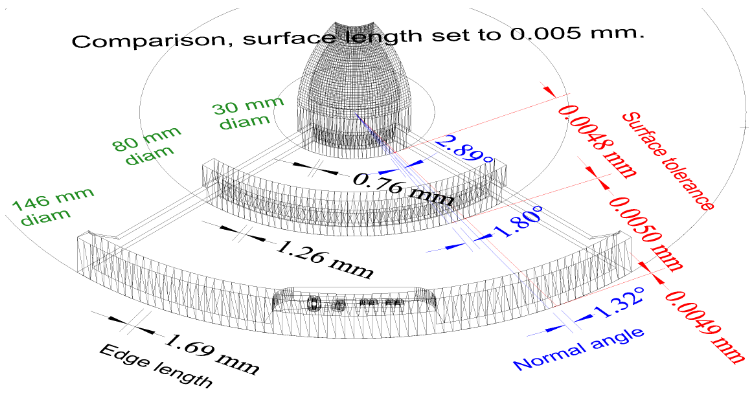


Fig 60, uses a surface tolerance of 0.005. as can be see, the surface tolerance remains close to the set value, on the three different diameters.

The Normal tolerance is set to 45 (to prevent it interfering), and an edge length set at '0'.

The image shows that the Normal and edge tolerances, automatically change value depending on the diameter (and radius) of object. This is necessary so that TC can attempt to keep the surface tolerance as close as possible to the set value of 0.005.

Normal Tolerance.

From the help file. '' Specifies the maximum angle (in degrees) between surface normal's at points on a facet. ''.

The normal angle at the default 15 degrees is a little course for 3d printing. Being roughly equivalent to setting the main facet quality slider to '54' And will show as facet lines if printed.

With the standard quality slider set to 100 in the ACIS dialog, it is equal to approx 4 degree normal angle Fig 62. Therefore if this is good enough for ones printing, there is no need to use expert faceter, simply set the standard quality slider to 100.

One time the quality may require 'expert' settings is if the curved object is quite large. Being based on angles, the further away from the centre of the arc, the greater the facet edge length, and thus the more faceted 3d printing.

Reducing the Normal angle (tolerance) to 2 gives a good finish on medium objects, but again may not look as good on large items,

Fig 63. Looks very good on an object of 18 mm diameter, with an edge length of just 0.3 mm. Which is too small really, but carried out here to show the effect.

Fig 64

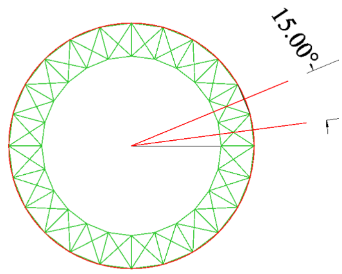


Fig 62

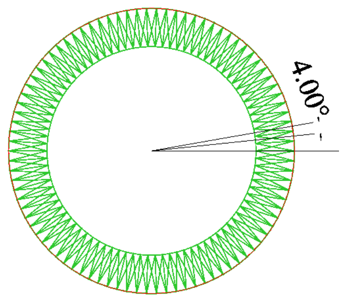
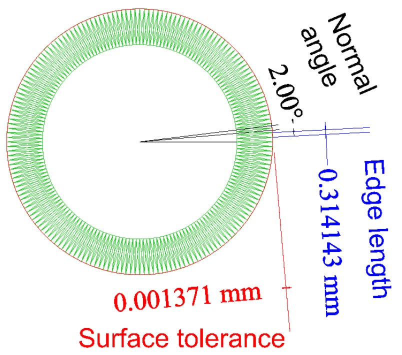
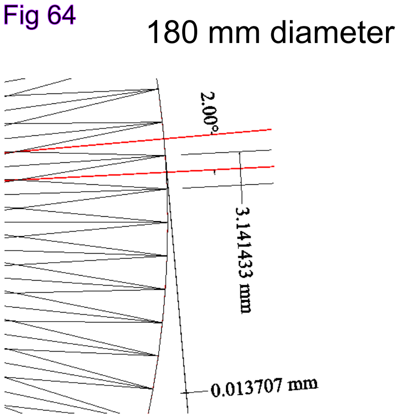


Fig 63 18 mm diameter



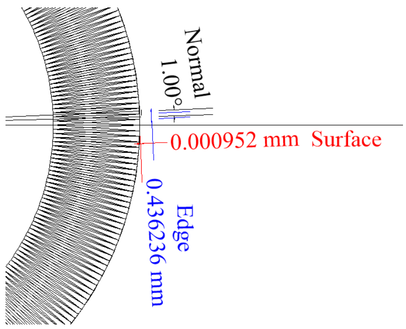
However the same tolerance on a 180 mm diam object, Fig 64. Gives an edge length of 3.1 mm which may be fine for many prints, but may need reducing if the printer shows some faceting.

Fig 64



The minimum setting for Normal tolerance is 1. This should be fine for most sized objects. Fig 65. Shows a normal of 1 with a 50 mm diam object, the edge length of only 0.44 mm Which although a bit too fine, it should provide a good finish.

Fig 65



For anything finer than this we would need to use one of the other settings, like surface tolerance or edge length.

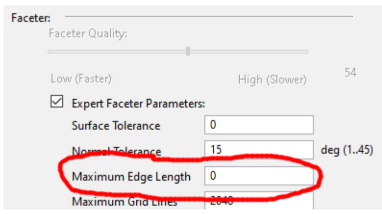
Fig 45 on page 11, shown the changes to surface tolerance and edge length, when the 'Normal tolerance (angle) is set, with the surface and edge set to zero.

Maximum Edge Length.

From the help file '' Constrains the maximum length of a side of a grid cell in object space. More precisely, the maximum edge length value constrains the maximum length of a diagonal of a grid cell, because the length of a grid cell diagonal provides an upper limit for the edge length of a facet. Therefore, this parameter also constrains the maximum edge length of a facet. ''

The default setting for edge length or 0. Which disable the setting and TC uses other settings like normal tolerance for the faceting.

Fig 66



The normal ACIS slider setting of 100, gives a variable edge length, as it is based on Normal tolerance. Therefore if you require a specific edge length, you need to use expert faceter settings.

Unlike surface tolerance, where we could go down to 0.0002 for example. We do not want to take it anywhere near that low with edge length. Not only could TC become unresponsive, but it would be a complete waste of time, as 3D printers cannot go down to that level of accuracy.

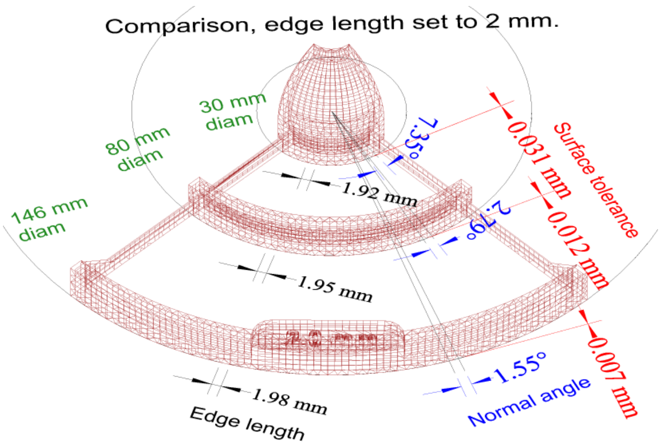
Indeed anything lower than about 0.4 edge length could be quite a waste, As previously stated, Edge length, surface tolerance and normal tolerance are interlinked, Therefore one would avoid setting surface tolerance and edge length, because one will override the other depending on the setting. The 'Normal' default tolerance of 15, gives an edge length of 2.35 on an 18mm object. So setting a low edge length will nearly always override the normal tolerance.

There may be occasion where you need to set the 'Normal angle to its max of 45. To prevent TC using the normal angle instead of the edge length.

Fig 67, shows the comparison between Normal angle and surface tolerance, with the edge length is set to 2mm. For different circle (cylinder) diameters.

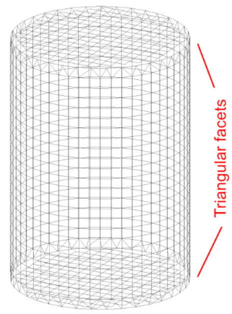
With larger objects, this is where edge length does come into its own.
TC attempts to keep the same edge length, and thus the same smoothness, irrespective of size, by varying the normal angle, and surface tolerance to suit. .

Fig 67



Using a too small an edge length on a large object can slow TC down, therefore may be best to change length on a copy of the file just before printing.

Fig 68

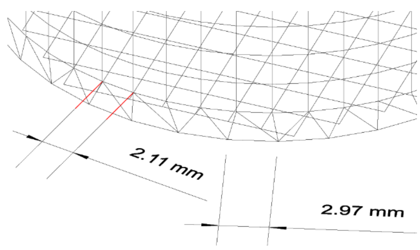


One problem that can occur if one is measuring the facets. When a cylinder is created, If the expert faceter is ticked, and one sets an edge length as the parameter, the bottom and top of the cylinder side can be triangular, whilst the rest of the body is rectangular.

This needs to be taken into account if one is calculating the facets, as the triangles may be a different size to the rest of the body.

Fig 69. Shows a difference of over 0.8 mm.

Fig 69



This may not seem much, but can mean the facets are of slightly less quality when 3D printed.

Grid lines.

From the help file.

‘’ Min Grid Lines U: Specifies the minimum number of u grid lines (i.e., grid lines in the u-direction).

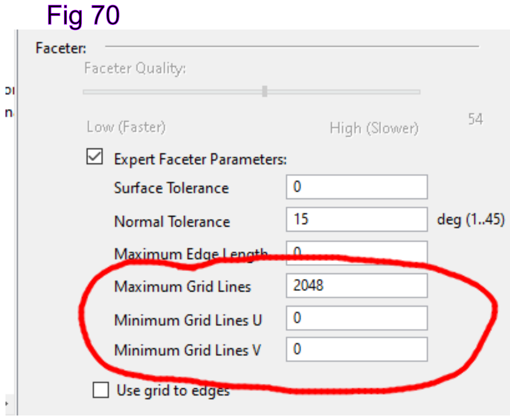
Min Grid Lines V: Specifies the minimum number of v grid lines (i.e., grid lines in the v-direction).

** The following grid aspect ratio may not be available in all TC versions **

Grid aspect ratio-Specifies the maximum ratio of the long side to the short side of a grid cell in 3D space. NOTE does not appear in some versions.

The aspect ratio value does not guarantee that triangles will have a particular aspect ratio; it applies only to the aspect ratio of grid cells. If the value of the grid aspect ratio parameter is less than or equal to 0.0, the grid aspect ratio is ignored. If the value of the grid aspect ratio parameter is greater than 0.0 and less than or equal to 1.0, a value of 1.0 is used. The default grid_aspect_ratio value is 0, which means the grid aspect ratio is ignored.

Fig 70



Use Grid to Edges: Specifies whether a grid is used and whether the points where the grid cuts the edge is inserted to the edge. ‘’

One may think that the ACIS solid would be like an mesh wrapped on the whole surface equally based on the minimum grid settings. However that is only half correct. Grid setting is used alongside one of the other settings, (surface, Normal or edge.)

The look, and the facets differ depending on multiple settings. The mesh around the object could be based on the surface, normal or edge setting, with the height and flat surfaces being based on the grid.. Below is a comparison of some settings associated with the grid.

The back of the object has been moved for clarity.

Fig 71

The first example Fig 71. is a basic reference image, without using the grid setting. It uses the default 15 for the normal tolerance.

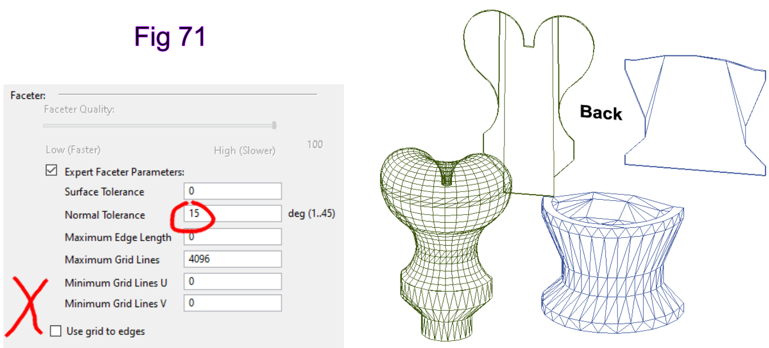
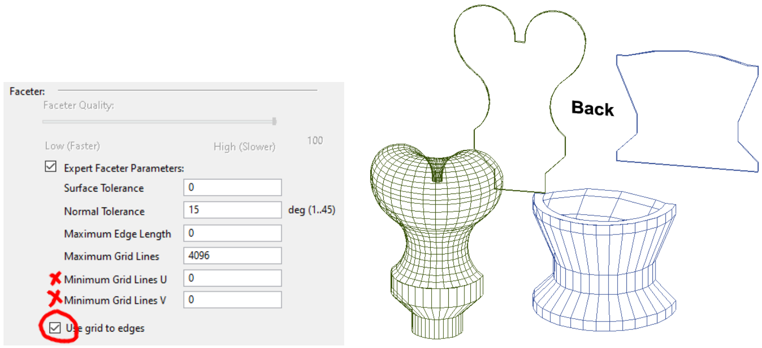


Fig 72. Uses the same 15 normal tolerance, but with ‘Use grid to edge’ turned on (ticked).

The result being, most triangular facets are replace with a rectangular grid.

Fig 72



Important note. When ‘Use Grid is ticked, it directly affects the other tolerances, ever if minimum U / V is left at zero. Exactly how it calculates this affect, I do not know. But in tests, the normal tolerance is reduced to approximately 0.7059 times the angle.

What this means is, Using the default normal tolerance of 15, and turning on Use Grid, the angle is reduced to 15 * 0.7059 = 10.5885 degrees. If the normal tolerance is set to 2. Then new angle after ticking Use Grid is 2 * 0.7059 = 1.4118 degrees.

Whilst the above affects the Normal tolerance. Edge length will generally be the same, with grid ticked or not ticked. As TurboCAD tries to accommodate the edge length one has entered.

However, as Surface tolerance, is dependent on Sine and Cosine. The calculation does not hold true for this setting. As a rough estimate, the surface tolerance alteration is generally between .49 and 0.5. Though in tests, this appeared to sometimes be inconsistent.

What this means is, With grid turned off. If we have a 30 mm cylinder, with surface tolerance to, say, 0.002. This equates to 1.8653 degrees normal, and 0.4883 mm edge. However ticking Use Grid, would change that to surface tolerance 0.001, Normal = 1.3235, and edge length to 0.3465. which may be too fine than what one actually needs.

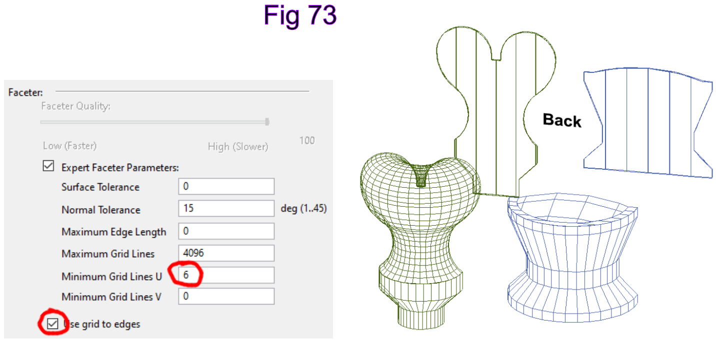
The reason for mentioning the above, is simply to show the relationship between the different settings. And the effect of ticking ‘Use Grid’ has on the other settings. I doubt anyone would actually wish to manually calculate all the different figures

Carrying on.

Fig 73. Shows the first alteration, by putting 6 in the Grid lines U box.

The effect applies the grid in the X direction. The reason for no change in the x grid spacing on the two revolves, is likely due to the 15 in the normal tolerance.

Fig 73



To show the grid U effect on the object. In Fig 74. The Normal has been set a maximum (45), so as not to affect the outcome too much, and the grid 'U' cranked up to 64.

Grid 'V' is left at zero, thus the grid is elongated. This example was simply done to emphasise the grid effect.

Fig 74

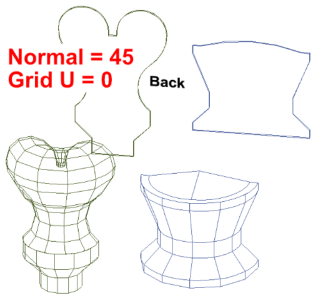


Fig 75

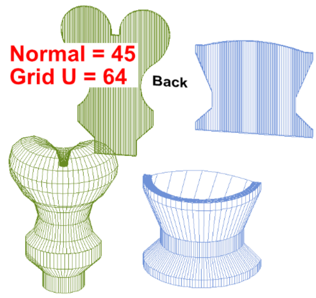


Fig 76. Shows the effect of just having the grid V set.

In this example, Normal tolerance was set at default 15, and grid V set to 6.

As can be seen. Grid V affects faces going in the Y (up) direction. Grid U is set at zero, so the X direction lines are set using the normal tolerance value.

Fig 76

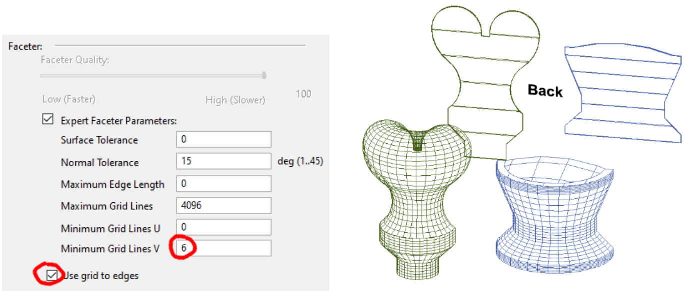
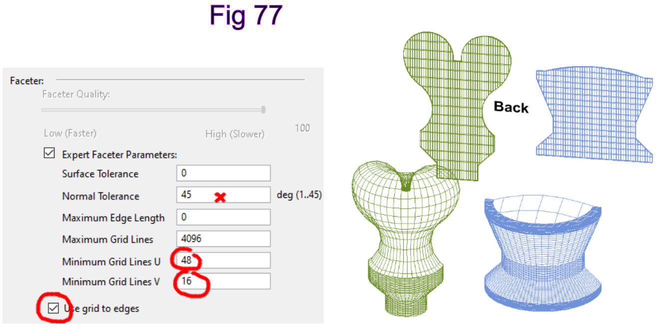


Fig 77. Shows the effect of setting both the grid U and grid V to a positive figure.

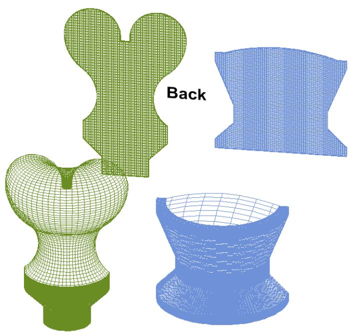
Normal tolerance is set to 45, thus keeping the normal tolerance course, allowing the grid U setting to affect the outcome. Grid U is set to 48 (64 is the maximum allowed). Grid V set to 16.

Fig 77



This will result in a smoother 3D printed object.

Fig 78



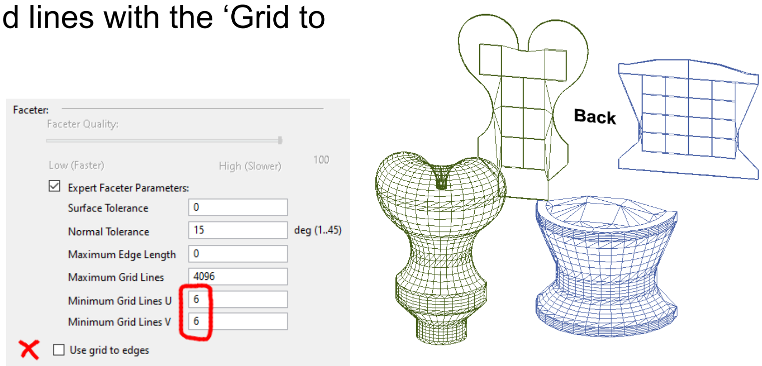
It may be tempting to increase grid U and grid V their max of 64. As shown in Fig 78. However this is not necessarily the best solution, because the amount of facets it produces.

It is pointless having so many facets, that it produces a drawing facet height of 0.025mm, and printing from the slicer at 0.2 layer height.

The last example simply uses the grid lines with the ‘Grid to edge’ turned off.

This retains some of the triangular facets, and as can be seen on the back, restricts the grid to flat areas fully within the object. This can mean that some areas may not show the grid.

Fig 79



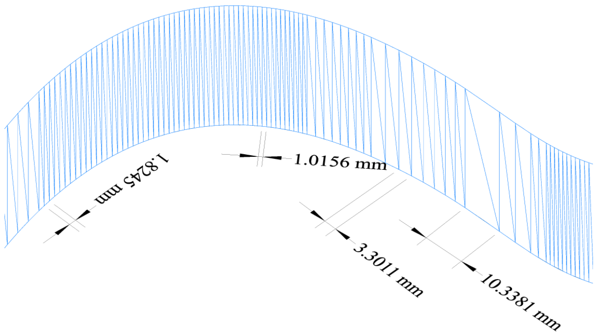
For 3D printing, it is a balance between number of facets, the physical size of the object, and what the printer can deliver.

Spline / Bezier.

As stated earlier, splines and bezier will generally have variable facets, when extruded or revolved.

As can be seen in Fig 80. There are facets from 1 mm to 10 mm, depending on the curvature.

Fig 80



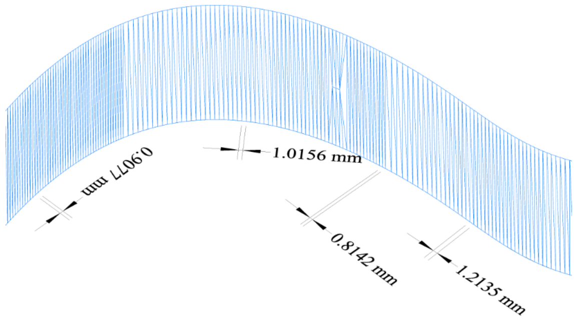
This makes it difficult to keep the whole object at a consistent smoothness on large objects. Fig 80. Uses an expert faceter, Normal tolerance = 2.0 degrees.

We can to change some other settings to get more consistent facets

We can often improve things by choosing a low surface tolerance.

Fig 81. Uses a surface tolerance of 0.005. Normal tolerance was left at 15 degrees. As can be seen, this is a big improvement over normal tolerance.

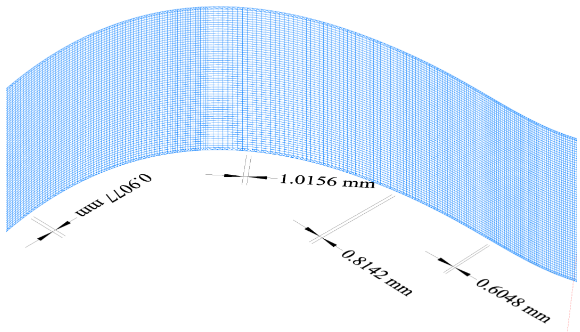
Fig 81



Using Maximum edge length is similar to surface tolerance, but can restrict any shallow curves from keeping large facets.

Fig 82. Uses an edge length of just 1.5 mm. Producing a fairly consistent smooth curve.

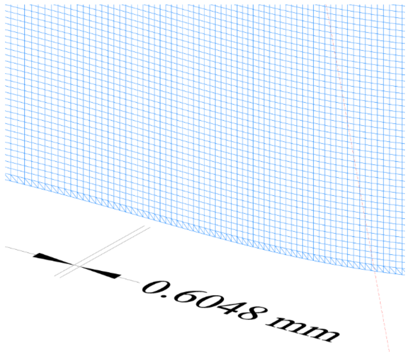
Fig 82



The reason that it looks quite different to surface tolerance (Fig 81.) is due to the mesh that is created when using edge length Fig 83. The mesh can be both vertical and horizontal.

This can give an optical illusion that the facets are much smaller than surface tolerance.

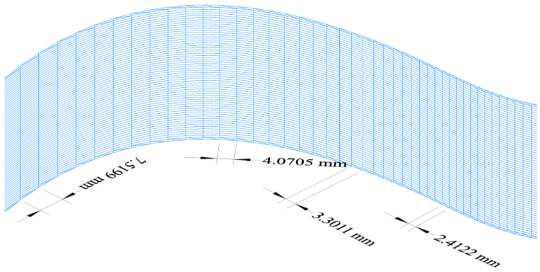
Fig 83



Grid settings can also be used with spline / bezier curves, However, on its own the results can be poor.

As can be seen in Fig 84. Using the same spine curve as previously, facets are as large as 7.5 mm, using grid U and V set to the maximum 64.

Fig 84

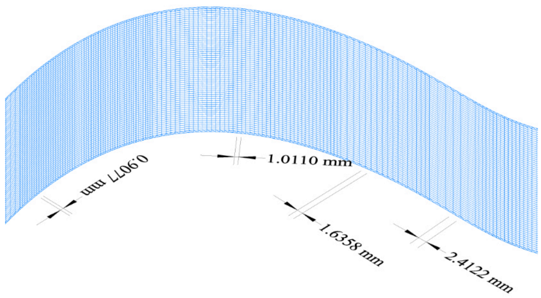


On object like this sheet curve, a good use for the grid, is to combine it with surface tolerance, which often gives a rectangular mesh instead of the usual triangular mesh normally associated with surface tolerance.

Fig 85. Shows the use of Grid at maximum 64, and Surface tolerance of 0.004.

This still has the odd larger facet of 2.4 mm, but the majority should produce a smooth 3d print.

Fig 85

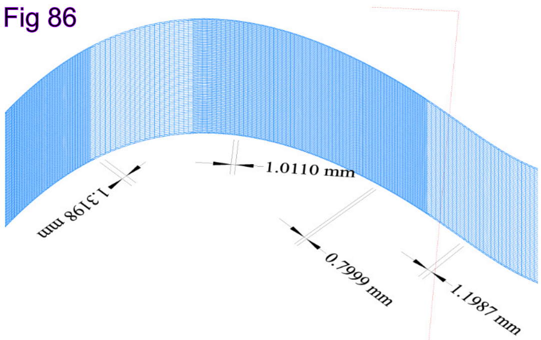


Using grid with edge length works quite well.
In Fig 86 Grid was set to 64. The Edge length was set at 1.5 mm. Normal was at the default 15.

Although there are still some variations in the facet sizes, all facets are within the 1.5 limit we have set.

When compared with Surface tolerance (fig 85.), Edge length can in some circumstances, keep the facets within a closed range.

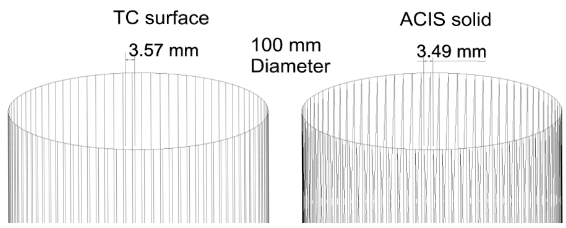
Fig 86



What about Smesh. (TurboCAD Platinum).

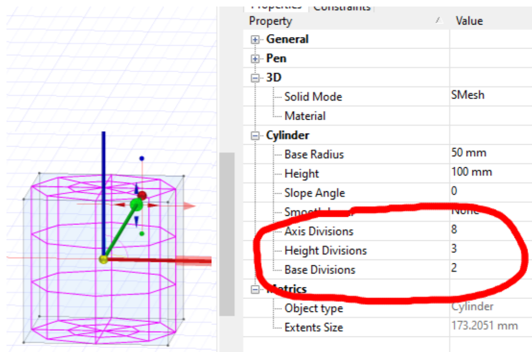
Smesh is TurboCAD's organic modelling arm. It is similar to TC surfaces, but allows for variable amounts of curvature to be added.

Fig 87



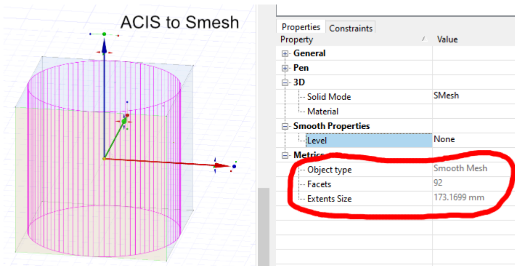
With TC surfaces and ACIS solids, the facets are that will be printed, are predictable. As shown in Fig 87.

Fig 88



With Smesh, the faceting depend on how the object is created. If the object is a native object, (as labelled in the selection palette). i.e. sphere, cylinder etc. The facets are manually set when an object is converted to Smesh. Note spin, loft etc. are not native objects. Fig 88.

Fig 89



If the object is labelled as ACIS or TC surface, the facets are taken from the original object. With the selection palette reporting the number of facets converted. Fig 89.

This is not the place to discuss the operation / modelling using Smesh. We are only concerned with the output for printing.

Fig 90

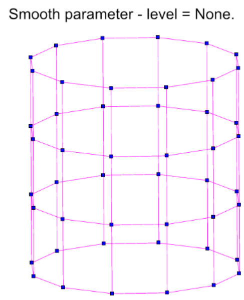


Fig 90 shows the object edit node selected with the smooth level set to 'None'. This has few facets, as we only set low divisions as in Fig 88.

This is obviously not good for 3D printing. However, we can easily change that, by increasing the smoothness level.

In Fig 91. The object as given a crease on the top and bottom faces. The level was change to 2. However simply changing the lever does not change the facets of the object.

Fig 91

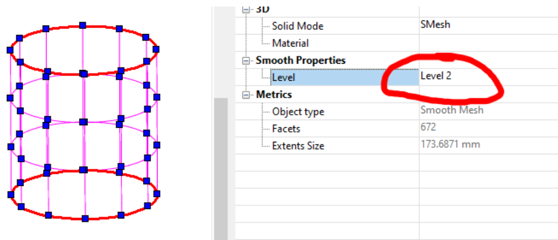


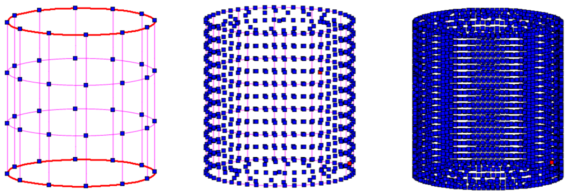
Fig 92. Has the Smesh exploded once into a TC surface.

When exploded it takes on the facet count, appropriate to the selected level. Automatically creating the appropriate number of nodes.

Fig 92

Exploded to TC surface.

Level None Level 2 Level 3



The effect also occurs when saving an object to STL etc.. TC creates the necessary nodes / facets when saving the file. This keeps the original object as an Smesh for further refining.

In this instance, a 97 mm diameter cylinder, equates to 2.39 edge length on lever 3. Therefore should print quite well.

Smesh Conclusion

What one sees in wire frame, in the TurboCAD drawing window, is not generally what will be saved for 3D printing (regarding Smesh).

Because the Smesh has a simplified mesh displayed, which is a good thing. It interpolates nodes when saved for stl etc. (or exploded), depending on smooth level. therefore this needs remembering when saving in a non-TC format.

The easiest method is simply to duplicate the object, and explode once. Select the copy with the edit tool, and take a look at how the nodes are displayed. Then delete the copy.

Smesh, also has the ability to add new edges, refine fact, split face etc. to manipulate how the object looks. Which in tern affects the output for 3D printing.

File formats.

The format that has been used with 3d printers for many years is STL. This format was adopted by the 3D community as the standard file format, from the start of RepRap program.

These days many other formats are available, with STEP starting to become a popular format. However not all slicers support STEP, also deluxe version of TurboCAD cannot save-as STEP.

Other files that are sometimes used include OBJ, 3MF, and X3D among others. (TurboCAD cannot save as X3D)

TurboCAD deluxe users have a limited choice, basically STL would be the only viable option for a 3D model prior to 2022. V2022 deluxe can also save as 3MF.

Platinum versions have slightly more choice, STL, OBJ, 3MF, STEP (and STP) would be suitable. SVG is possible but rather odd.

STL. (stereolithography). Deluxe, Pro, and Platinum

STL works quite well. A majority of the time, STL's can be sent straight from TurboCAD into the slicer, Slicers these days are quite powerful, and generally won't bother about things like reverse normal's.

There are the odd occasion where an STL from TC will have 'bad geometry'. And the slicer warns of problems. However if that occurs, one can open the STL in a third party 'free' program. Let the program repair the part. Save and open in the slicer.

STL files are unit-less, meaning 1 unit in TC is equal to one scale unit in the slicer, irrespective of what that unit is. For example, scale units in TC are set to metres, draw a box 10 m x 10 m x 1 m. Save as STL. Open in the slicer and it will be 10 mm x 10 mm x 1 mm. This is because STL does not know size, only that it is 10 scale units. Some slicers may have a way to preset the imported STL units , like setting the units to inches instead of mm. Older slicers may not have that function.

What the above means, is the user needs to ensure the units are correct in TurboCAD to match what is needed by the slicer.

One drawback with STL, is using multiple parts. If multiple parts are saved in the same STL. They are treated as one object, even if they are not touching. Therefore, unless the slicer has a 'slice' or separate function built in, the one cannot move individual objects around the slicer print bed.

3MF.

A more modern format (2015) than STL (1987). And thus can incorporate more information than an STL. Including colour and multi-material support.

3MF has the ability to save multiple objects in a single file, and allow the slicer to position then individually on the build plate.

This means, if you have an assembly, of five parts, one can export the model, and rearrange the parts in a slicer for printing. Rather than having to save five different STL files.

3MF will export colour information. Though whether a particular slicer will use this information is unknown.

One problem I encountered with 3mf, using an older version of a customised (by the printer makers) cura, was that the model imported into the slicer outside the build area. It appears the import coordinate system would not use object extents in positioning the part/s. The only way round this was to reset the model in TurboCAD, so that it was positioned on X=0, Y= 0, Z = centre of extents of the model, meaning that the bottom of the base as at Z = 0.

This then allowed the correct opening in the slicer. This oddity, did not happen with a new version of cura. However the new version imported the parts separately onto the build plate, without asking first. Whereas a version of Prusa slicer asked if I wanted a single model or separate parts.

This is just to show, not all slicers are equal, and things may need tweaking in TC. Depending on your slicer requirements.

OBJ

Cannot be tested on my computer.   OBJ saved from TC, crashes both Cura and a version of Prusa slicer. Examining the part in a third party mesh program, showed faults.

Therefore my advice is either avoid saving as OBJ. Or test immediately after saving if it will open correctly.

STEP. (and stp)

Step and STP are the same format. Even though they are listed separately.

Step is another format that can have multiple objects saved within one file. Allowing the slicer to position them individually if needed. Step will also export colour information.

An advantage is the file size, these can be greatly smaller than the same file saved as STL.

Cura (v5.7.2) cannot open Step files. This test was done using a version of Prusa slicer (customised by a printer manufacturer). Orca slicer can also import Step. (I have not tried all the slicers).

SVG.

SVG will save and open in some slicers. However SVG is a 2D format, which can result in some odd prints.   However large SVG may not open in the slicer. Fig 93. Shows an SVG opened in a slicer, and looks like a 3D file. However looking from the side Fig 94, shows the Slicer, simply extruded the 2D SVG to give it thickness. And then warned that it had created too many triangles.

Fig 93

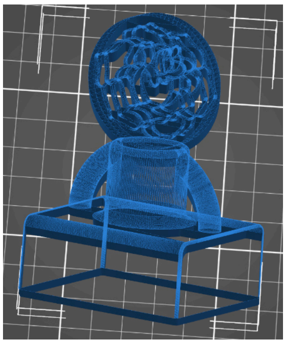
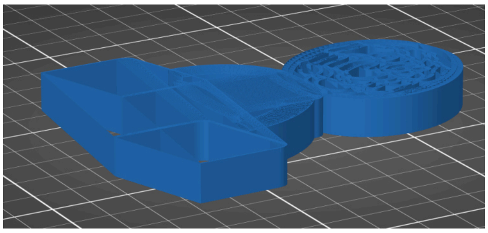


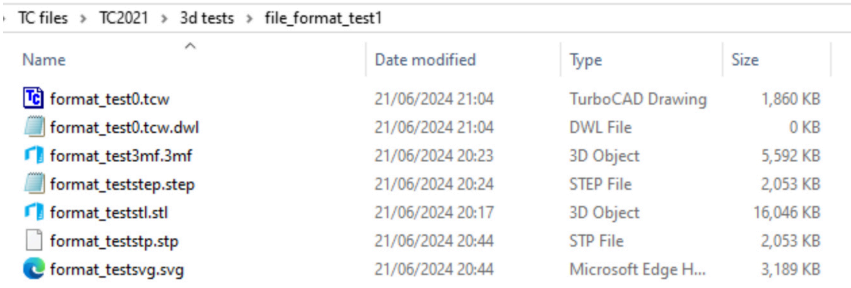
Fig 94



Comparison

Fig 95. Shows the main file types, and the size on disk. Because STEP /STP is vector based the file size is far less than the triangular mesh of the STL. The triangular STL can produce large files.

Fig 95



TC files › TC2021 › 3d tests › file_format_test1

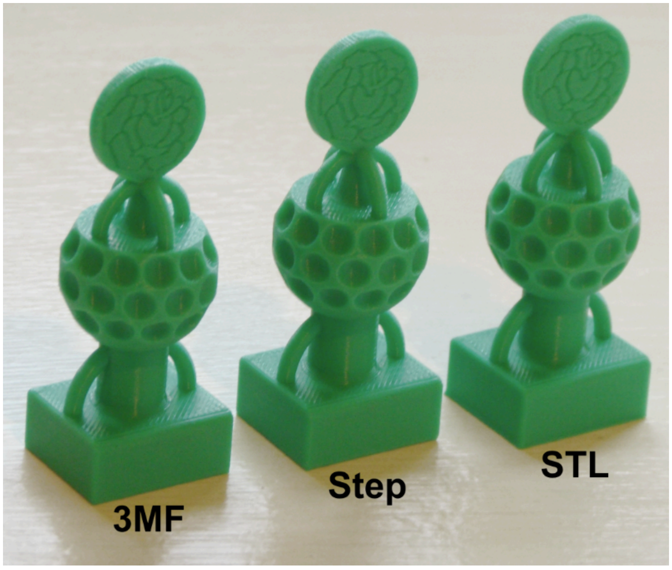
Name	Date modified	Type	Size
format_test0.tcw	21/06/2024 21:04	TurboCAD Drawing	1,860 KB
format_test0.tcw.dwl	21/06/2024 21:04	DWL File	0 KB
format_test3mf.3mf	21/06/2024 20:23	3D Object	5,592 KB
format_teststep.step	21/06/2024 20:24	STEP File	2,053 KB
format_teststl.stl	21/06/2024 20:17	3D Object	16,046 KB
format_teststp.stp	21/06/2024 20:44	STP File	2,053 KB
format_testsvg.svg	21/06/2024 20:44	Microsoft Edge H...	3,189 KB

Finally, STL, 3MF and Step files were sliced and printed, just to see how the slicer handled the different formats. These were only small models,

There was very little difference between the three models.

The only problem area, was where the flower cylinder connects. However it was the same for all three formats, suggesting a printer problem rather that a file type problem

Fig 96



.

Is there a generic ‘Best’ way for 3D printing.

Unfortunately not. Printers differ, slicers differ, TurboCAD versions differ etc.

For TC surfaces.

Always set the number if approximation lines (in native object properties) to 88 (or 90 if you prefer). Before doing any Boolean operations on a native object.

Or on symmetrical objects, model half the object, mirror copy, and Boolean add.

Use spline or bezier for curved objects, if a finer mesh is required.

For ACIS solids.

For small objects of, say less than 35mm diam, The normal ACIS slider, set to 100 should be fine.

For larger objects, one has a number of options. The figures below are based on an edge length of less than 2.5 mm. Examples ..

Set the expert faceter ‘Normal’ setting to 2, up to 75 mm diam or even 1, up to 130 mm diam.

Set the surface tolerance, 0.007 up to 100 mm diam, 0.004 up to 180 mm.

Edge length 2 to 2.5 up to 50 mm diam, edge 1.0 to 1.5 should be OK for most other sizes. 0.5 in special circumstance.

Grid settings. These require experimentation. Example setting is ..

Turn on grid to edge. Grid U and V at 32. Surface tolerance = 0, Normal tolerance = 45, Edge length = 1.5. This allows the grid to control part of the object, whilst the edge length controls the circumference.

However, as previously stated, printers and slicers differ, so what works for one person, not be appropriate for someone else, with a different printer.

Finally.

Throughout this document, we have been concerned with improving facet count to get a better 3D print.

However it needs mentioning, that not all objects / drawings need to be concerned about faceting. If one is always drawing miniatures, of fine scale models. Then there are few occasions when one would need to use expert faceting, ACIS slider at 100 should be fine.

Expert faceting would generally be needed for large diameter objects.

Choose a facet setting which will satisfy the needs, without going too fine. These is no point trying for an edge length of 0.02 mm, TC will become too slow, The slicer may complain of too many facets, and the printer cannot print at that tolerance.

Lastly. No amount of faceting or expert faceting, will remove layer lines from an FDM printer. Shallow features like the top of a dome, will show a stepped appearance, irrespective of faceting. Though finer layers like 0.1 mm or 0.05 mm, can reduce the effect.

Calculation between surface, normal and edge.

ON the off chance someone may interested in the calculated relationship, between Normal's, Edge length, and Surface tolerance. Below are some calculations.

Note, there are some prerequisite figures required, depending on what one is trying to find.

A common figure for all calculations is Radius. Other figures are required depending on what one is trying to calculate. Refer to Fig 38, page 10. For diagram.

Find Edge length from Normal. Requires Radius and Normal Angle.

=(SIN(RADIANS(Angle/2))*Radius)*2

Find Surface from Normal (one of the following). Requires Radius and Normal Angle.

=Radius-(SQRT((Radius^2)-(SIN(RADIANS(Angle/2))*Radius)^2))
OR
=Radius-(Radius*(COS(RADIANS(Angle/2))))

Find Normal from surface tolerance. Requires Radius and Surface tolerance.

=2*(DEGREES(ASIN(SIN(RADIANS(90))*(SQRT((Radius^2)-((Radius-Surface tolerance)^2))/Radius))))
OR
=2*(180-90-DEGREES(ASIN((Radius-Surface tolerance)*(SIN(RADIANS(90))/Radius))))

Find Edge length from Surface tolerance. Requires Radius and Surface tolerance.

=SQRT((Radius^2)-((Radius-Surface tolerance)^2))*2

Find Normal from Edge length. Requires Radius and Edge length

=DEGREES(ASIN((SIN(RADIANS(90))*(Edge length/2))/Radius))*2

Find Surface tolerance from Edge length. Requires Radius and Edge length.

=Radius-(SQRT((Radius^2)-((Edge length/2)^2)))

If turning the grid on (with U and V set at zero) then. (see page 17).

If using Normal tolerance, Multiply the normal by 0.7059
If using Edge length, no change
If using Surface tolerance, multiply surface tolerance by 0.45 (This may or may not be accurate)