GEARWHEEL DESIGNER AND HOW IT INTERACTS WITH FUSION

Gearwheel Designer (GWD) is a software tool to design clock wheels. It was written by Graham Baxter and is relatively intuitive to use and has a good help file. There are quite a few variables that can be tweaked to achieve the exact result you need from a given wheel. This note is not intended to be a tutorial on GWD but more about how GWD can interact with Autodesk Fusion and with a 3D printer slicer program.

I have a paid licence to use Fusion and I have a friend who has the free to use hobbyist version of Fusion. We both have similar interests in our uses of the Fusion but often have been caught out on how different the functionality is between these two versions.

We both use GWD to design clock wheels. Once a design is complete the GWD allows us to export the design as a DXF, STL/AMF file and in GCode format. (It also has a geometric Points Curve export as a TXT file. This is aimed at use in the RS Design Spark Mechanical drafting software. This is a very specific format and probably not of general interest).

What follows is my perception and experiences interacting between GWD and Fusion. This might not be technically precise but hopefully will disperse some of the fog. I will try to identify the subtle differences between Fusion Hobbyist licence and a Fusion paid licence.

DXF Export

A DXF file is a 2D drafting file format that I believe was created by Autodesk. A DXF file of a finished clock wheel design exported from GWD will have a flat 2D sketch of the wheel. The sketch will be dimensionally correct. In order to manufacture the wheel this file needs to be extruded in depth to create a 3D model of the wheel. Once a 3D model is available this can be used for 3D printing the wheel or for creating a CNC program to manufacture the wheel on a milling machine. Importing a DXF file in this manner is the easiest route to create a 3D model from the GWD software.

Licenced version of Fusion – there is a direct import facility into the Fusion workspace for any DXF file. The DXF is 'Inserted' onto the Fusion workspace via the Insert sub menu in the Solid work area. Once in the work area it can be extruded to a thickness as desired. Modifications to the extruded 3D model are then possible in order to change aspects of the model such as inserting additional holes in the centre hub or changing the wheel spokes etc.

Hobby version of Fusion - the DXF import facility in the hobby version of Fusion is blocked.

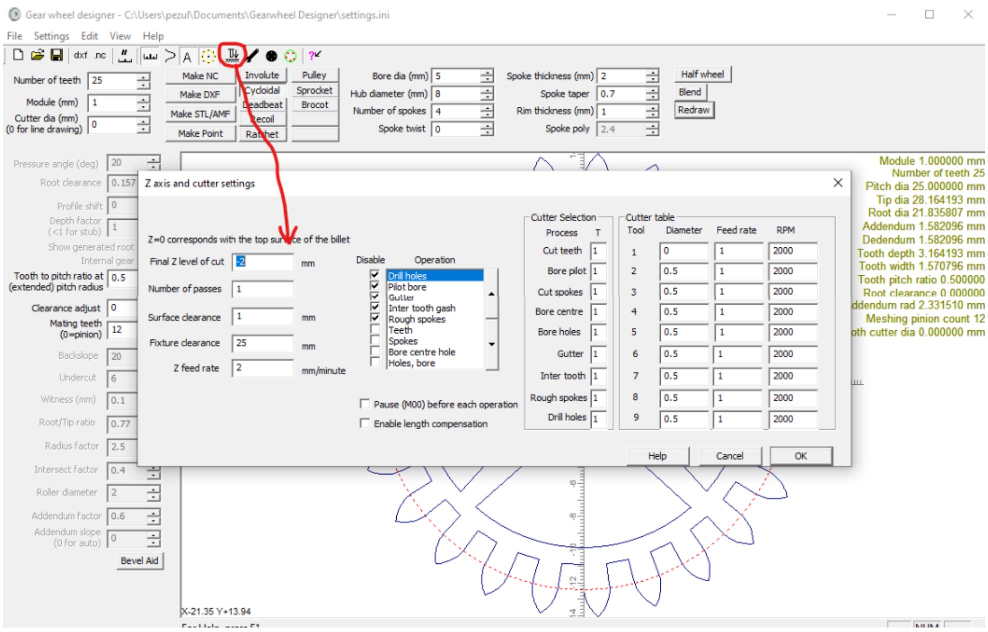
STL Export

This method uses the STL export function in GWD. It offers an alternative route to creating a solid 3D model for those with just the free hobby licence.

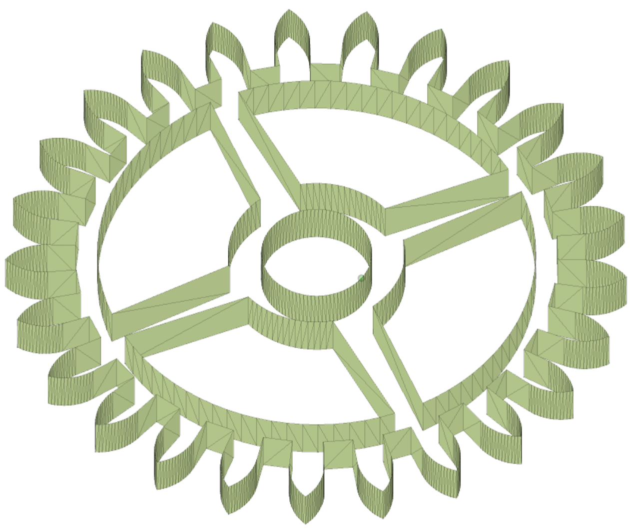
An STL format file is a way of saving a complete 3D model. In GWD the 2D drawing as mentioned above will need to be extruded to create a 3D version. This is done via the Z adjust function in GWD.

The value is entered as a negative value so if a wheel needs to be 3mm thick then a figure of -3 is entered in the Z adjust dialogue. Once the wheel has thickness it becomes a 3D model and can be exported using the GWD STL export function.

Would that things were that simple.



There is a shortcoming in GWD that although it allows a 3D extruding of the wheel design it does not extrude as a solid model. Instead it extrudes all the 2D sketch drawing lines by the Z adjust value but does not create any infill between these boundary lines. In short - solid borders but not a solid inner. It looks like a skeleton wheel as shown below.



Let's step back at this point.

STL files are used in 3D printing but a STL file will not directly drive a 3D printer. The STL file has to be sliced into layers and the layer geometry is deposited one layer at a time by molten plastic to build up the model. Think of it as starting off with a first layer like a crust of bread and then slowly building up slices one at a time on top of the bottom crust until the top crust is put in place and the loaf completed.

The slicer software is very powerful and juggles many factors in creating the control of the head that deposits the molten plastic.

If the STL file from GWD is imported into the slicer software the image created in the slicer working window will reflect the same skeleton image of the extruded walls of the model with no infill. If this could be sliced ready for sending to the printer it would just be lots of stringy sections of the walls of the STL with no internal body slices to bond them together. Not very useful.

There is however some magic that takes place. During the slicing process the slicer software intelligently decides that the walls should have infill and creates the geometry to do this. (Don't ask me how it knows how to do this…). The result is a solid model made up of the extruded walls but now with extruded internal mass between the walls. Because this is now a solid model the slicer can cut it into slices and pass them to the printer head to print full plastic layers.

The magic process is not quite perfect.

While GWD has defined how high the walls have to be, it does not pass any information regarding the infill. This is derived by the slicer software. The slicer software has an idea how tall the infill needs to be to match the walls but this is not a precise process and the printer while expecting to create a nice smooth final layer finds itself with stringy sections of plastic that are not quite finished. The top surface of the print is therefore likely to not be quite perfect but you might struggle to see this with the naked eye. The slicer software knows this imperfection is likely to happen and will flag it up as errors before you hit 'print'. The errors do not seem to inhibit the print and the printer does the best it can to finish the print.

In summary we now have a means of taking the GWD 2D drawing of the wheel and export it as a STL file into a 3D printer slicer software. The slicer software will heal the print's missing information and produce an accurate wheel albeit with some imperfections on the top surface.

If all you require is a 3D printed wheel there is no longer a need for Fusion as an in between step between GWD and the slicer program.

Editing the 3D Model

Problems arises if you have a need to modify the geometry of the wheel in aspects that GWD is not able to accomplish. For instance suppose the centre hole in the wheel needs to have a square hole rather than a round one. (This is not an option currently available in GWD).

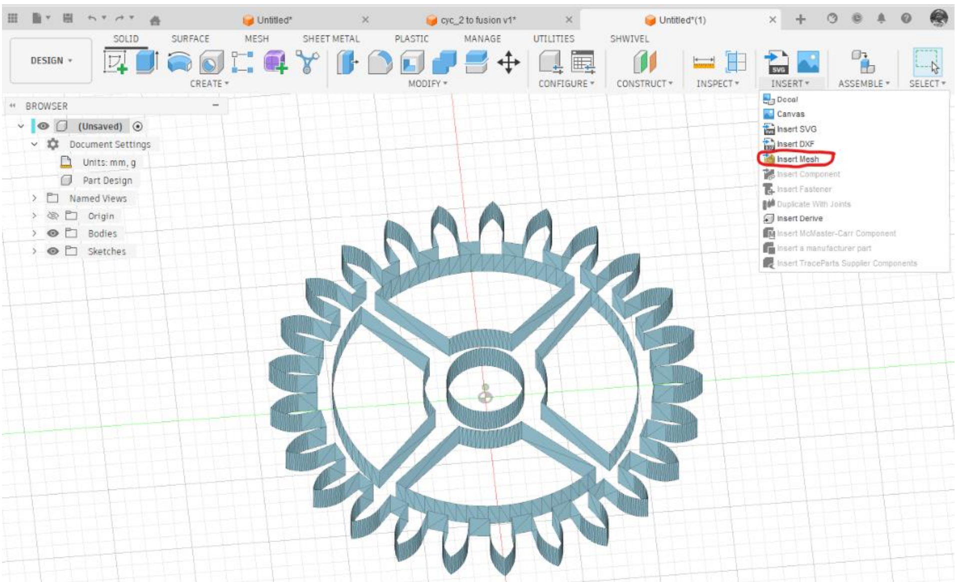
In order to make this type of fundamental change to a wheel design it would be beneficial to be able to import it into Fusion and edit it in a 3D modelling environment. A paid licence would allow this using the DXF import method but a hobbyist licence is blocked from the DXF import route.

Instead of using the DXF file from GWD, the STL file is imported into Fusion instead. This will be as a mesh import. This will display the same skeleton 'walls only' image with no infill. Fortunately there are functions in Fusion to overcome this and which can create the infill to produce a fully editable model of the wheel. These functions are applied differently depending on whether you are using a hobby version or a licenced version of Fusion. Both routes use the Mesh manipulation tab in Fusion.

The licenced version route is more complex and this will be detailed first.

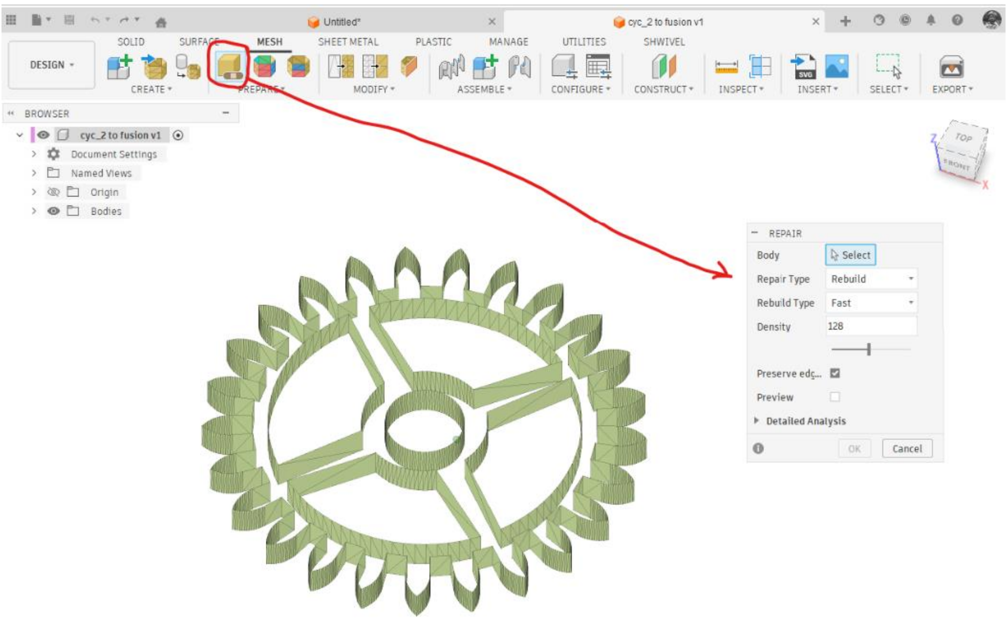
It is important to make a distinction here. The STL file is not uploaded as other files might be but rather Inserted as a Mesh. A STL file upload loses its dimensional information. A STL file opened using the Insert Mesh keeps this information. In the Solid work area go to Insert and select Insert

Mesh and select the STL file. You will now see the same skeleton form of the wheel as before and if you check its dimensions it should match the GWD model.



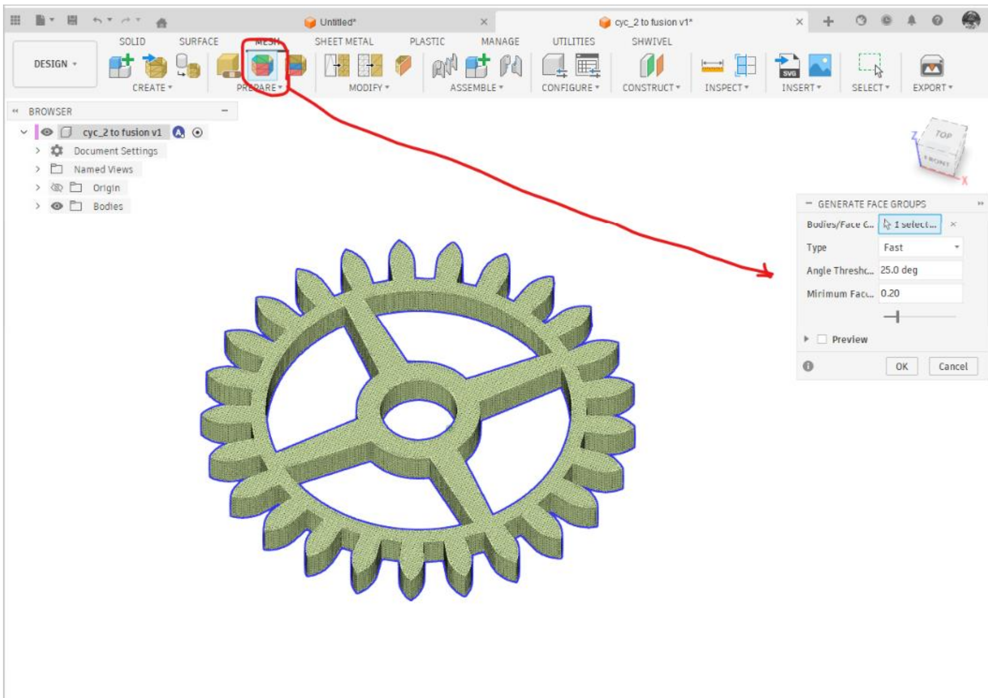
Zoom into the skeleton wheel and notice that the shape is made up of triangles (sometimes called facets). This is how 3D models are constructed. Any shape can be broken down into triangles. The first step is to 'repair' the STL model to have infill. Go to the Mesh tab and click on the Repair icon as shown below. This will bring up the sub menu as shown.

In the sub menu match all the selections as shown and make sure you tick the Preserve Edges box. You can also tick the Preview box to see what happens as you change parameters.

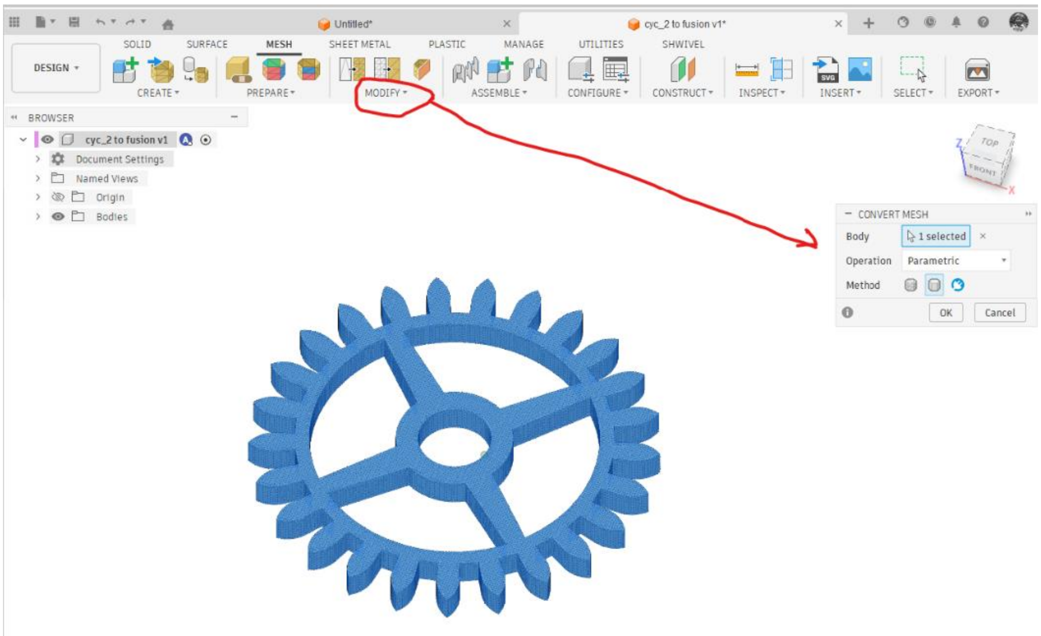


The result of the Repair will be an infilled model which has changed colour to a light blue. If you zoom in closely you will see the number of triangles has gone up dramatically.

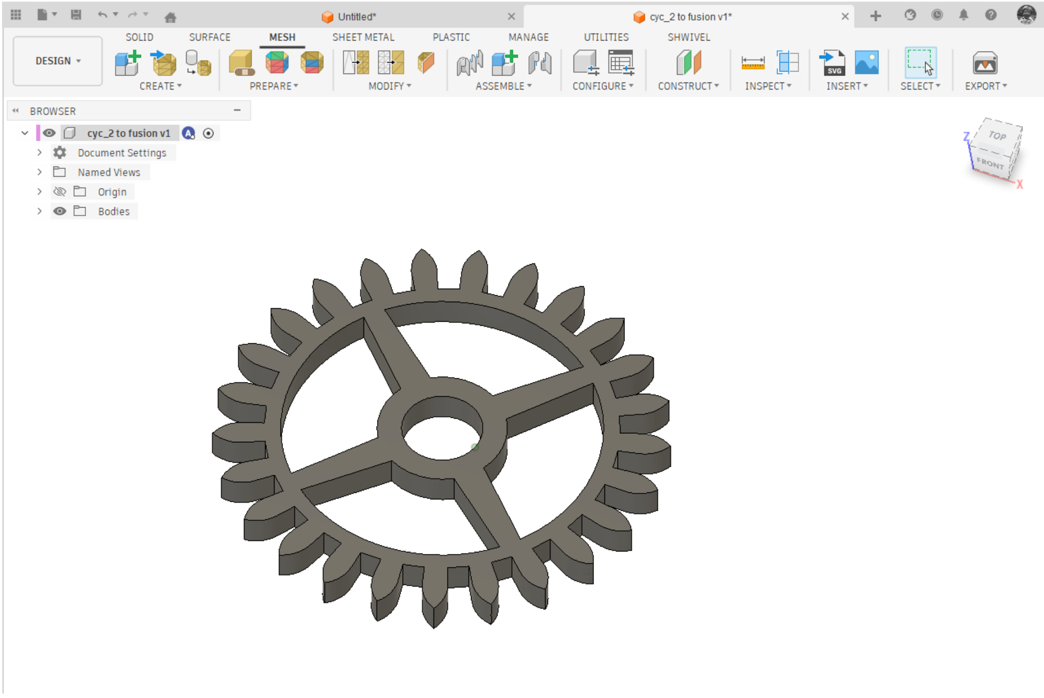
The next step is to Generate Face Groups so that like groups of triangles are brought together.



The transformed image will show colour in some of the wheel areas. The mesh is now ready to Convert to Solid. This function is found under the Modify tab. Set the parameters as shown and then OK. You will almost certainly get a warning that the model has in excess of 10,000 triangles and could take some time to compute.



After a short while you will have a solid version of your wheel as shown below. This can be sent to a 3D printer with all the repairs already completed and with no need for the slicer to do anything other than prepare the wheel for printing.



This completes the STL wheel importation and solid conversion using the licenced version of Fusion. The solid model is fully editable to add or modify features. If you need to create a CNC program from the solid model then go to the Manufacturing work area and create your setup and toolpaths as you normally would.

The hobbyist conversion is a simplified process. I cannot show screen shots as I do not have a hobbyist version available. The essence is that you follow the same initial route of inserting the mesh using the GWD STL skeleton walls file. Once on screen it will not show any triangular structural shapes. Now run the same Repair process as described in the licenced version and immediately the model will jump some of the above steps and the model will be converted to the solid model as per the last step in the licenced version.

This simplification by the hobby licence means there is no flexibility available to the hobby user in the Mesh conversion processes.

Ironing

There is one further useful thing to mention. Most slicers have an 'ironing' function. This option can be tick selected to be programmed into the slicer code sent to the printer. It causes the print head to go back over the top finishing layer of plastic and reflow the surface to 'iron out' lumps and bumps. This is not a perfect process but can reduce the top surface imperfections mentioned above when using the direct route of wheel of GWD to slicer printing rather than via Fusion.

Notes

This write up is based on Fusion 2606.1.22 and on GWD version 3.149

All 3D printing related issues are based on the Qidi slicer which is a skinned version of PrussaSlicer.