Exporting CAD Files for Rapid Prototyping

The preferred file format for CAD files is STL, which is an export function in most CAD software. Providing a “fixed”, “clean” STL file will make all the difference to the finished product; improving accuracy, efficiency and reducing cost – won’t incur fixing / reformatting costs.

Modelling Advice for Rapid Prototyping

It is important to bear in mind that parts will be rapid prototyped from the initial design/modelling stages, saving time and energy later on.

- All surfaces must be solid (have some thickness), closed and continuous.
- The default or advised tolerance/resolution etc in each software package according to the size of the model usually works well.
- If the model is designed to scale (the dimensions of the final rapid prototype model), fewer errors from scaling will be generated and the .stl file is likely to be “cleaner”.
- For moving parts allow a gap >0.5mm. (Please note this is a general value may vary depending on faceting).
- Design the mesh according to complexity of the surface; for example the back of a hedgehog will require a much finer mesh than that of a beetle.
- Avoid making enclosed empty areas.
- Check for duplicated surfaces created when offsetting parts; delete these before exporting .stl.
- The file size often reflects the errors in the file; normally a good binary .stl file is 10MB (although there are always exceptions).

Some packages offer additional advice in their respective *Help* menus for designing parts for Rapid Prototype, which is worth looking at.

Below are some recommendations for producing clean STL files from 3D designs in various CAD packages:

Magics

- Run the *Fix Wizard* to automatically correct file errors.
- If this does not work, some errors can be corrected manually by thickening walls, filling holes etc.
- Go to *File*, then *Save As* and select STL option in *File Type*.

Rhino

- Each part of the model should be a closed, solid polysurface.
- This can be checked using various commands, for example:
    - *SelOpenSrf*
    - *SelOpenPolysrf*
    - *ShowEdges*
- Open surfaces can be closed or capped and open edges can be joined.
- Then Mesh the object (may need to select all before meshing).
- Run *CheckMesh* command; this will produce a list – all meshes should be “good”.

- Run *UnifyMeshNormals* command.
- Export Mesh as STL. The *Simple Export Settings* are perfectly acceptable for meshing.

AutoCad

- The model should be a 3D solid.
- The value of *FACETRES* should be 10.
- Run *STLOUT* command.

Solidworks

- Go to *Tools*, then *Options* and select *Export*.
- (Alternatively go to *File*, then *Save As* select STL option in *File Type* and click *Options*).
- A dialogue box will appear for *Export Options.*
- Under *Output as* select *Binary* for a smaller file.
- Set *Quality / Resolution* to *Fine.*
- Then Save.

Pro/Engineer

- At the initial stages of design, set the accuracy; the default value of 0.0012 is normally sufficient, though for larger parts a larger value is advisable. (Please refer to the Pro/Engineer *Help* menu for more information.
- Go to *File*, *Export, Model* and click *STL Menu.*
- A dialogue box will appear; when prompted for the *Chord Height,* insert the value 0.01 and click *Enter*. (The remaining defaults in this menu should be sufficient).
- The *Chord Height* may change; if so, the *Angle Control* must be adjusted as follows:
    - *Chord Height between 0.01 and 0.02 → Angle Control = 0*
    - *Chord Height between 0.02 and 0.04 → Angle Control = 0.5*
    - *Chord Height between 0.04 and 0.05 → Angle Control = 1.0*
- Click *OK.*

Euclid

- Use the *exchangeb* utility to export native data from Euclid.
- Use *Create .Stl* Menu to generate the file.
- If this is not easily re-computed polyhedral polygons can be used to create the file.
- Set the *polygonisation coefficient* to 2.0 (the finest resolution)

Ideas

- Go to *File, Export, Rapid Prototype File.*
- Select the model / part
- A dialogue box will appear; use the settings below
    - *Prototype Device = SLA500.DAT*
    - *Part Positioning = ORIGIN*
    - *Absolute facet deviation = 0.01*
    - *Output file type = Binary*
- Click *OK*

Unigraphics

- Go to *File, Export, Rapid Prototype File.*
- Use the settings below
    - *Triangle Tolerance = 0.0025*
    - *Adjacency Tolerance = 0.12*
    - *Normal Display = OFF*
    - *Triangle Display = ON*
    - *Output type = Binary*
    - *Auto Normal Gen = ON*

Mechanical Desktop

- Run *AMSTLOUT* command
- The following settings affect the file in the following way:
    - Reduce *Angular Tolerance* (default = 15^{o}) → increase resolution
    - Change *Aspect Ratio* (default = 0 – this ignores the option) → change height : width facet ratio (*Aspect Ratio* = 1 → height = width)
    - Change *Surface Tolerance* (default = 0 – this ignores the option) → change greatest distance from edge of facet to actual geometry
    - Change *Vertex Spacing* (default = 0 – this ignores the option) → change length of a facet edge.

Solid Edge

- Go to *File, Save As* and select STL option in *File Type*.
- Click *Options* (in dialogue box)
- Use the settings below:
    - *Conversion Tolerance* = 0.0254mm / 0.001”
    - *Surface Plane Angle* = 45^{o}
- Click *Save*

Solid Designer

- Go to *File, External* and *Save STL*
- Choose *Binary* mode
- Select *Part*
- Type in the filename
- Set *Max Deviation Distance* = 0.01mm
- Click *OK*