Procedure For Creating Useable Solid Models from Scanned (.stl) Files

General

When irregular objects are scanned to .stl files, the resultant model is a 3-D triangular element mesh reproducing the scanned shape in much the same way as a geodesic dome approximates a hemisphere. The more irregular the shape, the harder it becomes for the triangular elements to fully match up to form a perfect representation of the original object. This is true because the triangular elements which make up the scanned model want to be equilateral triangles of the same size. When the modeled body has small holes, protrusions and sharp jagged edges, it becomes impossible to match up the edges of all of the elements to form a perfect mesh without seriously distorting the elements, which automatic scanners tend not to do. Instead of failing, the scanner generates the best mesh it can, but mesh faults still remain.

The object of this effort was to produce a solid model of some type (.stp, .step, iges) from the scanned .stl mesh in order to use a solid model CAD system of some type to analyze, measure and document the original scanned object. In particular, a CAD system was to be used to create an orthographic drawing of archaeological points showing relevant dimensions, surface area and volume of each.

In this particular case, SolidWorks™ was the CAD system used, but since generic solid models such as .step files can be imported into all solid model CAD systems, any such system can be used. While .stp files can be opened and displayed by SolidWorks™ and other CAD systems, they are a graphical representation only and do not have features which can be selected for dimensioning, etc. For this reason, the .stl files have to be converted to .step (solid) files.

Conversion Software

One problem with manipulation of fine mesh .stl files is that they are very large (many elements) and hard (time-consuming) to manipulate in CAD systems. For example, 5931_LS_0005.stl, is a mesh with 353,212 facets (triangles). Therefore, it would be desirable to reduce the number of mesh points without losing any dimensional information. An open source program which does this is MeshMan. MeshMan or Mesh Manipulator is a 3D modeling tool which performs function called "Decimate" which reduces the file size by reducing the number of facets in the mesh. It was found by experimentation that the best procedure for using this function is to divide by 2. Otherwise, the program tends to crash. MeshMan can be downloaded (free) from www.holmes3d.net/graphics/**meshman**/ .

A problem with MeshMan is that it will not directly open a .stl file. It requires an .obj file. Therefore we need a program which opens a .stl file and exports a .obj file. One such program is

MeshMixer, available for download from http://www.meshmixer.com/download.html . It then allows one to export that file as a binary .stl file which will then be further processed.

In the process of reducing file size by dividing the facet count by 2, 4, or 8 a number of new mesh faults are introduced. One good program for repairing those meshes is FreeCAD. This open source freeware is available at http://www.freecadweb.org/wiki/?title=Download . It is recommended that every time a file is divided by 2, it is opened in FreeCAD and the mesh analyzed and repaired. Use the pull-down menu in FreeCAD to select "mesh design" and then use the menu "meshes" to select "Analyze", the "Evaluate and Repair". Continue using "Analyze" and "Repair" until no more mesh errors are indicated. When the mesh is as clean as FreeCAD can fix, export the file as a binary .stl file.

Once the original .stl scan file has been decimated and repaired, the final piece of the puzzle is software designed specifically to convert .stl files to .step (or .stp) files which are generic solid models. Software which does this is InStep available for download at https://www.solveering.com/instep.htm . This software is not free, but is very inexpensive. A license for file operations only, that is, conversion of .stl files to .stp files is $25. More expensive licenses are available for "design" ($75), and "analysis" ($160) but are not necessary for our objective here.

Import .stl files which have been decimated and repaired into InStep, and export them as .stp files. These files still have mesh faults, but can be opened as solid or surface models in FreeCAD or SolidWorks™. InStep also calculates model surface area and volume.

Detailed Conversion Procedure

1. Use MeshMixer to import original .stl scan file.
2. Export file from MeshMixer to .obj file.
3. Open .obj file with MeshMan.
4. Under the 'Tools' Menu, select 'Decimate'. Input a target mesh size equal to one-half the original count, or one less if the number is odd. Experience shows that decimation of more than 1/8 destroys the original file.
5. Export the decimated file as a binary .stl file.
6. Open a 'New' FreeCAD file.
7. Import the decimated file as a .stl file.
8. Select 'Mesh Design' from the main drop-down menu.
9. Select 'Analyze/Evaluate and Repair' from the 'Meshes' menu.
10. Iterate 'Analyze and Repair' the file until no more mesh problems are indicated.
11. If the file shows multiple 'bodies', select bodies one by one, change color to determine which body is the main body, and delete the minor ones. This may result in what looks like tiny holes in the main model. Repeat evaluate and repair after minor bodies are deleted.
12. Export the file as a binary .stl file.
13. Import the file into InStep V2.3.
14. Export the file as .stp file. Note surface area and volume if desired.
15. Open .stp file in CAD system of choice and create dimensioned drawing.

Results with Point 5931-LS-0005

The file 5931 LS 005 was used to investigate solid modeling techniques. For reference, the original .stl file size was 17.252 MB. After conversion to .stp file, the original file size was 37.505 MB. This .stp file would not load into SolidWorks™ with the workstation running out of memory after running overnight and unsuccessfully trying to heal all mesh faults.

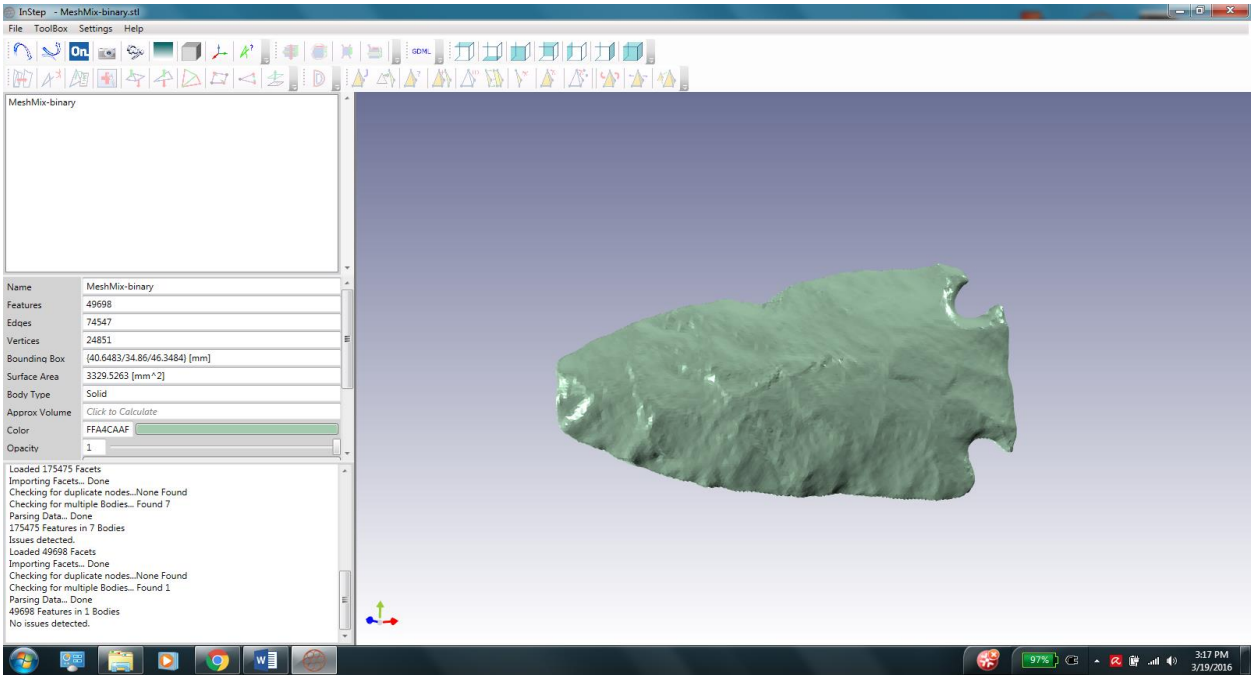
The file was then decimated by 2, 4 , 8, and 16 per the procedure in the previous section. Resulting file sizes were:

FILE	.stl INPUT	.stp OUTPUT
DecimateBy2	8.628 MB	137.255 MB
DecimateBy4	4.018 MB	63.098 MB
DecimateBy8	2.032 MB	35.832 MB
DecimateBy16	1.016 MB	0.318 MB

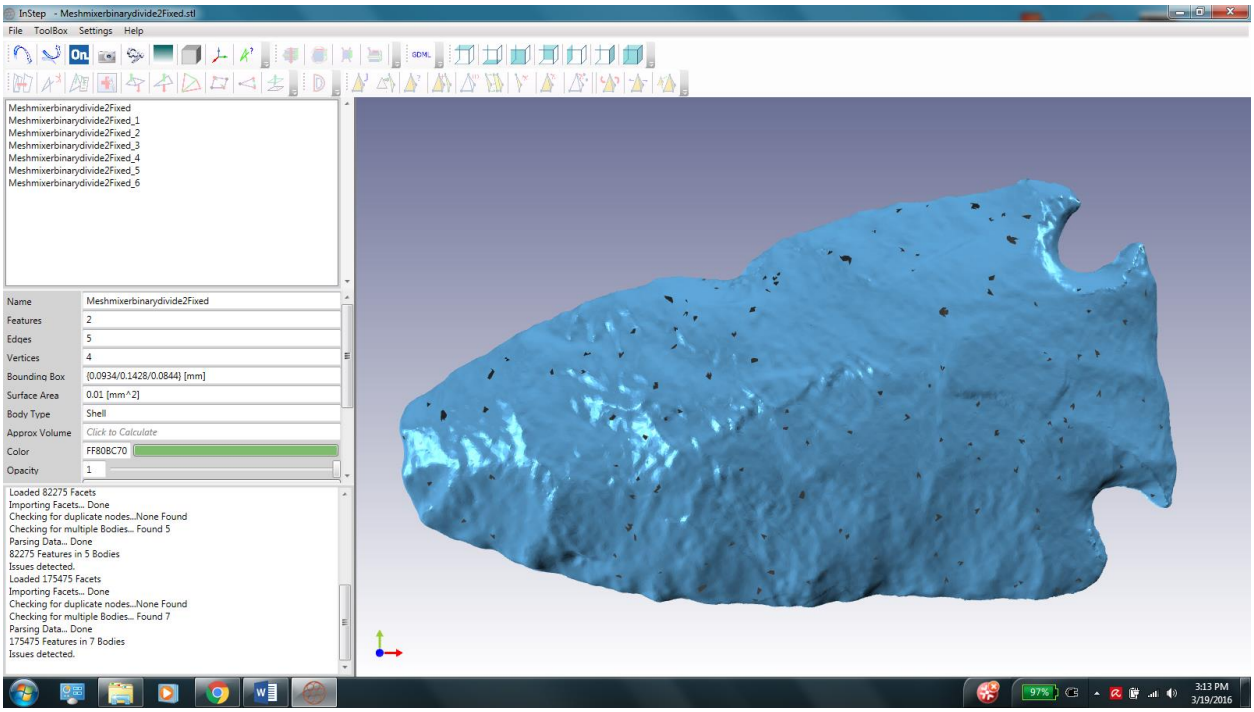
The DecimateBy16.stp file contained no useful solid information and when opened in SolidWorks™, showed as a few tiny disjointed fragments.

The following five InStep V2.3 screenshots show the .stl input files of the original file, the DecimateBy2, DecimateBy4, DecimateBy8, and DecimateBy16.stl files to demonstrate that the basic dimensional information after decimation is not adversely affected. The shots are prior to export as .stp files.

Original .stl file

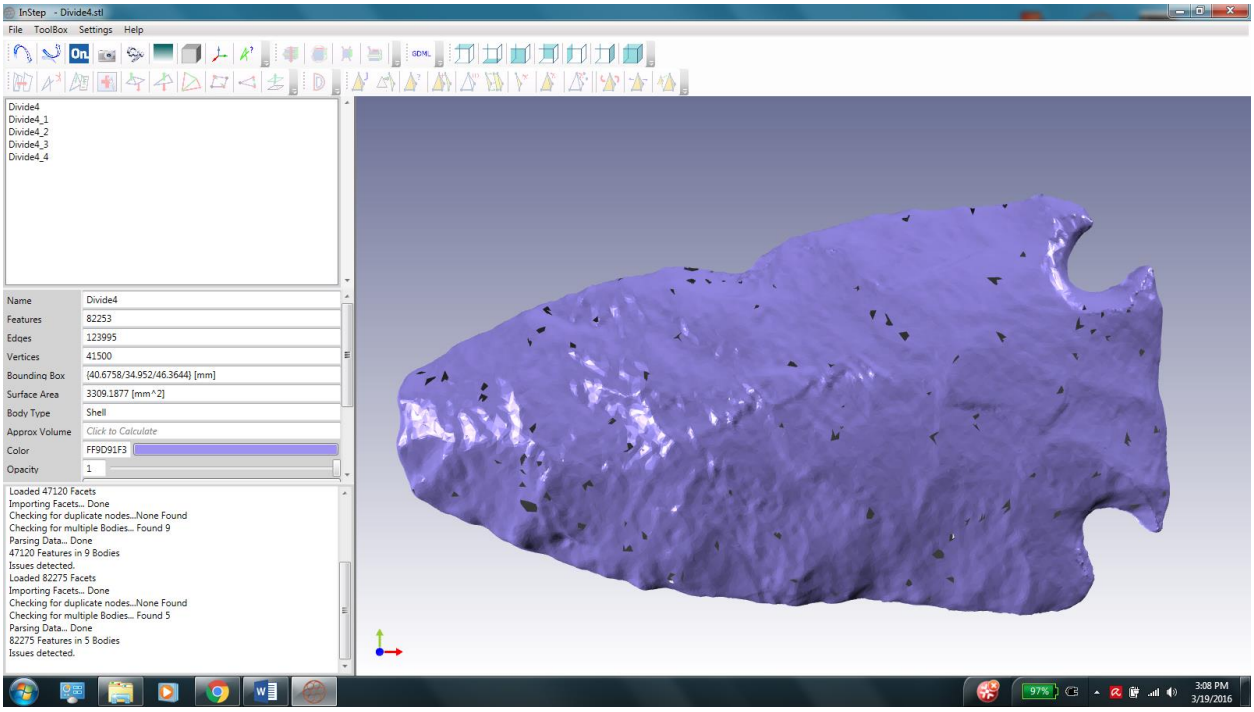


DecimateBy2 .stl File



Note black spots where facets have been lost during decimation and repair.

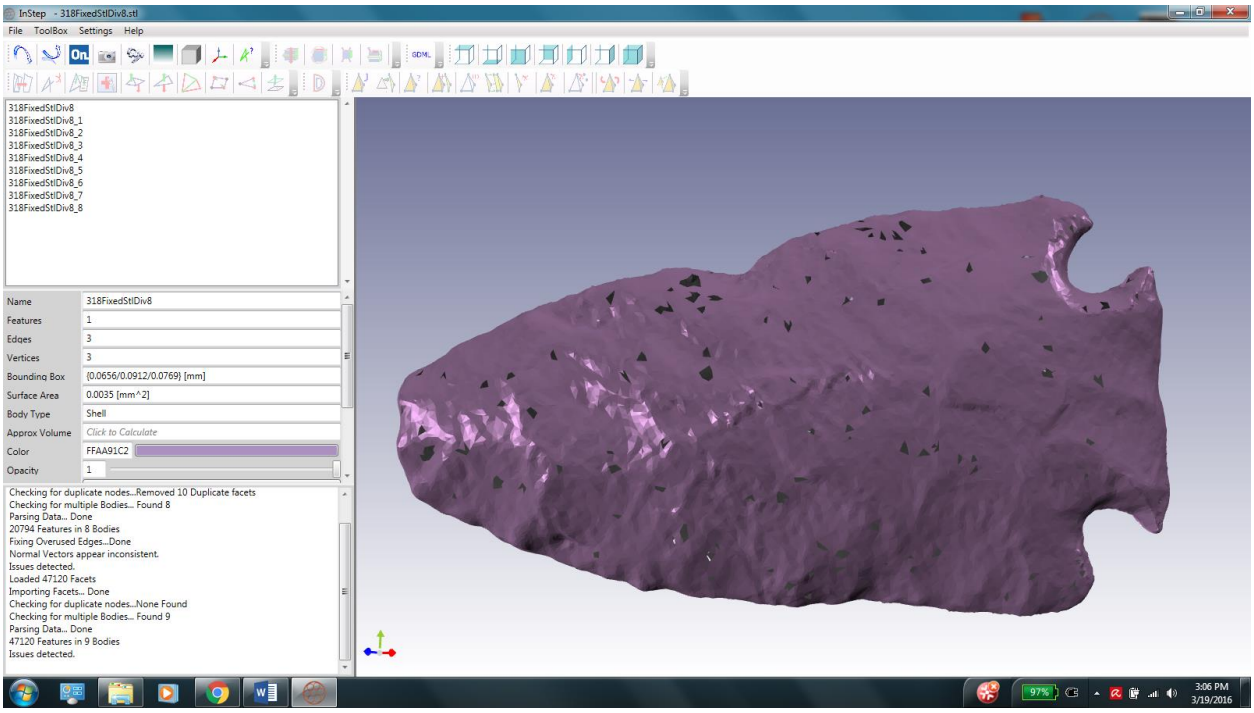
DecimateBy4 .stl file



DecimateBy8 .stl file

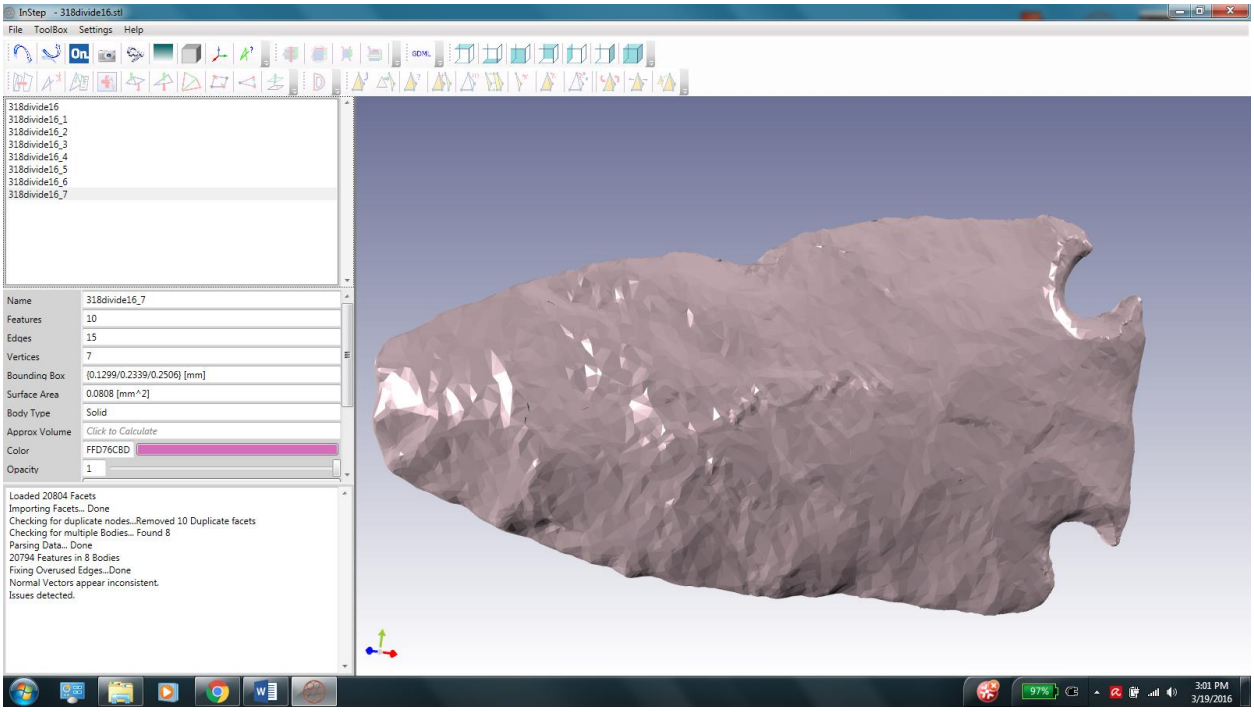
This file converted to .stp file and used in

SolidWorks™ to generate output drawing.



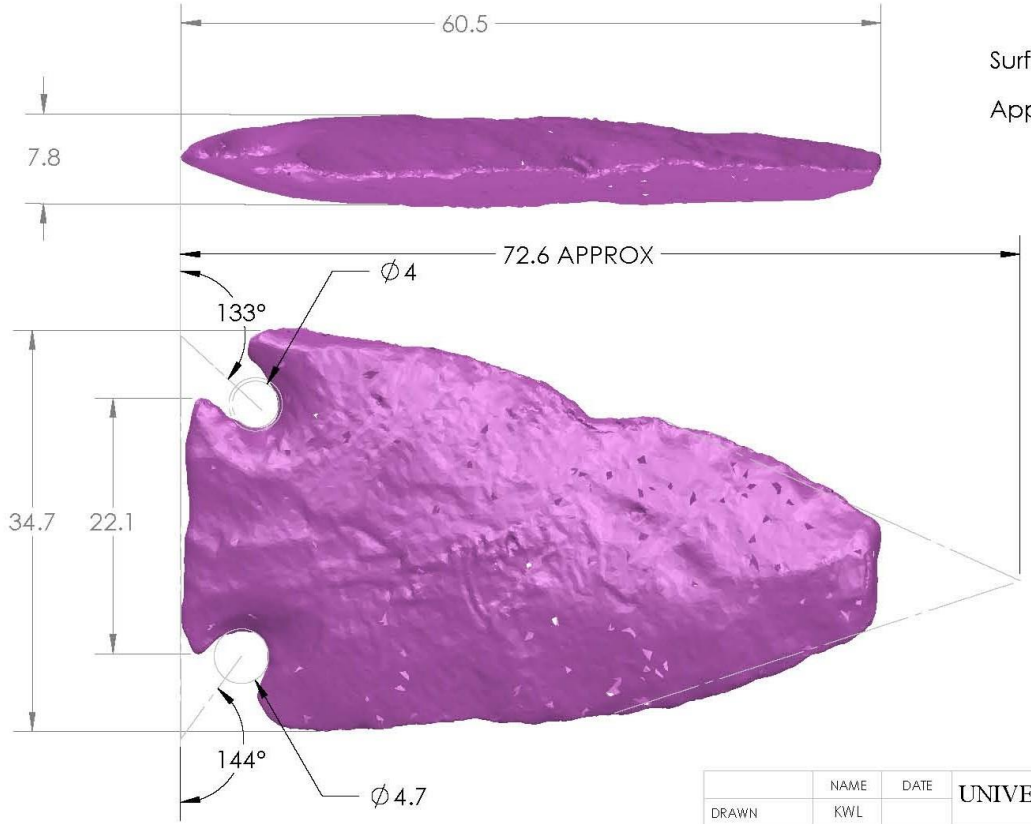
DecimateBy16.stl file

(Not used but .stl file is still dimensionally viable. Note coarser facets on model.)



The DecimateBy8.stp file was opened in SolidWorks™ 2013 and oriented so an accurate orthographic drawing could be made. Since the XYZ coordinate system of the scanning mechanism was different from the SolidWorks™ XYZ coordinate system, a reference plane was constructed through the point mid-plane by manually placing three reference (geometric) points equally spaced along the 'sharpened' perimeter of the Kirk point. The reference plane was then defined by these three points. This plane was then used as the 'front' view in the drawing. Establishing an accurate mid-plane and viewing the point normal to this plane is necessary to minimize parallax errors in dimensioning.

The drawing is shown below and sent as a separate .pdf document.



Surface Area: 3343 sq mm

Approx Volume: 6900 cu mm

	NAME	DATE
DRAWN	KWL	
PROJ MGR	AAW	

COMMENTS:

UNIVERSITY OF SOUTH CAROLINA

TITLE:

KIRK PROJECT DATA

SIZE	DWG. NO.	REV
A	5931 LS 0005	0

SCALE: 1:1 SHEET 1 OF 1

5 4 3 2 1

Conclusions

It is possible to generate orthographic drawings to document Kirk point dimensions. Those shown above are guesses as to what important. Estimated file processing time from scanned .stl file to dimensioned drawing in .pdf format is approximately 2-4 hours per file. After practice, the file decimation and conversion can be done in about 1 hour. Since dimensioning is done manually by picking mesh points, and the reference sketch geometry is unique to each Kirk point, another 1-2 hours is required for drawing prep.

The drawing format is arbitrary and needs to be defined. Once finalized, the format can be saved and used for all points.

SolidWorks™ is not the only CAD system which can open .stp files and generate drawings. FreeCAD probably has adequate capability, but will take some time to master.