

# DXF FEM Calculator 1.0

Josiah Wong – Updated June 2019

## Table of Contents

Introduction .....	3
Application Overview .....	4
Using the Application .....	5
Formatting a DXF File .....	6
Test Sample Results .....	8
Source Code .....	11

## Introduction

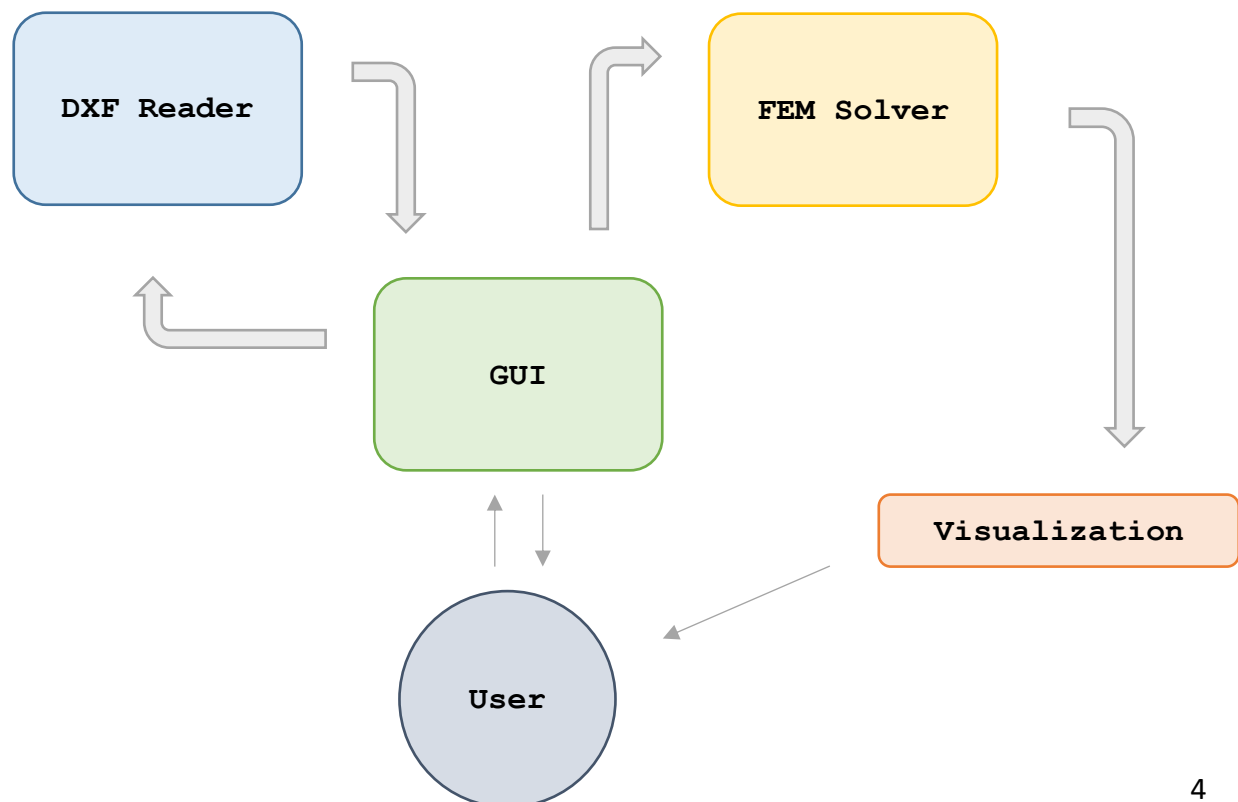
With the advent of modern computing, applying traditional engineering finite element method (FEM) to complex objects and systems has become increasingly rapid and accessible. Multiple commercial software packages currently exist to target specific areas such as heat transfer and load stress distribution, and vary in both size, scope, and complexity. However, few tools exist to address quick and modular calculations based upon pre-existing DXF drawings, a common file format utilized in communicating engineering design and parts specifications.

DXF FEM Calculator 1.0 proposes to bridge this gap, offering a rapid and iterative solution for AutoCAD-based DXF-file 2D plane-stress FEM calculations. By formatting pre-existing DXF files in a certain fashion, customized FEM studies may be run, which include specifying dimensions, material properties, and meshing fidelity. Further, parameter studies may be conducted to examine the impact of varying specific dimensions of the object geometry. While obviously limited in scope relative to other dedicated FEM software packages, DXF FEM Calculator 1.0 provides an easily accessible method to validate rough estimations before committing substantial resources towards more rigorous testing. For the source code, please see the *Source Code* section.

## Application Overview

DXF FEM Calculator 1.0 was constructed from various libraries in Python3 and has been modularized into three key modules: the graphical user interface (GUI), FEM solver, DXF reader, and visualization engine. By compartmentalizing each major aspect of the application in this way, further expansion and improvement of the application's various features may easily be implemented. Below, each section is briefly described:

- ❖ **Graphical User Interface (GUI)** – This is the module that controls the main window through which the user interfaces with the application. Inputs determining object geometry dimensions, material properties, and meshing fidelity can be specified here. Further, the GUI provides a method for saving templates, loading files, and toggling FEM solver results. The GUI was built using PyQt5 and constructed in the dedicated QtDesigner app.
- ❖ **FEM Solver** – This is the central FEM module that solves the stress distribution problem under plane stress assumptions. The resulting output includes displacement and Von Mises stress values for individual meshing elements, and can be viewed graphically similar to contemporary FEM packages. The FEM Solver utilizes the Calfem library for the majority of its core functionality, which in turn is built upon Gmsh and VisVis libraries for meshing and visualizing results, respectively.
- ❖ **DXF Reader** – This is the module that interfaces with DXF files, and whose functionality includes reading, interpreting, and manipulating the geometry found in a given file. Its chief function is to convert the DXF-formatted geometry into a form that is compatible with the FEM Solver. The DXF Reader relies on the dxfgrabber and dxf2svg libraries for parsing and displaying the DXF geometry.



## Using the Application

Using DXF FEM Calculator 1.0 can be both easy and straightforward. By following the below steps, a FEM calculation or study can be conducted:

1. **Prepare the DXF File.** In AutoCAD, create and format a DXF file that includes the object's geometry to be used for the FEM study. Please see the following section *Formatting a DXF File* for specific details. Once finished, save the file.
2. **Load the DXF File.** In the program GUI toolbar, click **File → Load DXF File** and select the DXF file created in Step 1. Conversely, if a previously-saved template would like to be loaded instead, click **File → Open...** and select the template to be opened.
3. **Specify Inputs.** In the program GUI, specify aspects of the FEM calculation. This includes object geometry dimensions, material properties, and meshing fidelity. Additionally, model output graphics can be toggled on/off. A parameter study can also be toggled which will run multiple studies using varied parameters as well. For more information, please type and enter 'help' into the GUI command prompt.
4. **Save the template.** (*optional*) If the current set of inputs (and reference to the DXF file) would like to be saved, under the GUI toolbar click **File → Save / Save As...** and select the filename and save location for the template to be saved.
5. **Execute the Calculation.** In the program GUI toolbar, click **Calc → Execute** to run the FEM calculation.
6. **View the Results.** In the GUI command prompt, a formatted list of model inputs and outputs should be displayed. If toggled on, the model output figure(s) should automatically be displayed as well. If running a Parameter Study, the outputted .vtk files can be viewed in the ParaView software program. For more information on viewing .vtk files, please type and enter 'help -vtk' into the GUI command prompt.

In general, information regarding all aspects of the program functionality may be described in further detail via the command prompt and can be accessed by entering 'help' into the command prompt.

## Formatting a DXF File

In order for the program to successfully interpret the DXF geometry, specific formatting techniques must be implemented within the DXF file. Below, the various features and limitations of potential DXF files are described:

- **Supported Drawing Tools**

Currently, only the **LINE** and **CIRCLE** tools are supported (with the exception of the rectangle / **LWPOLYLINE** tool to define the geometry frame). Further support for tools such as the **ARC** and **SPLINE** tools may be added in the future. Additionally, the **CIRCLE** tool may currently only be used for interior lines (i.e.: cannot be used to define the outer geometry).

- **Defining Exterior vs Interior (i.e.: Hole) Geometry**

To distinguish between exterior and interior geometry, the **LINEWEIGHT** options are used. Any exterior geometry must have a lineweight of at least **0.50mm**, and any interior geometry must have a lineweight less than that value.

- **Defining Individual Dimensions and Variables**

The **LAYER** feature in AutoCAD is utilized to distinguish specific geometry edge(s) to be used as boundary condition, force surface, or modifiable dimensions. To assign a variable to an edge(s), simply create a new layer and assign the edge(s) to the layer. The layer name is the variable name that can be called within the program GUI to apply a boundary condition, force, or dimension to the respective edge(s). It is recommend to apply unique colors and text labels for each specific layer to reduce ambiguity for the user.

- **Defining the Geometry Frame**

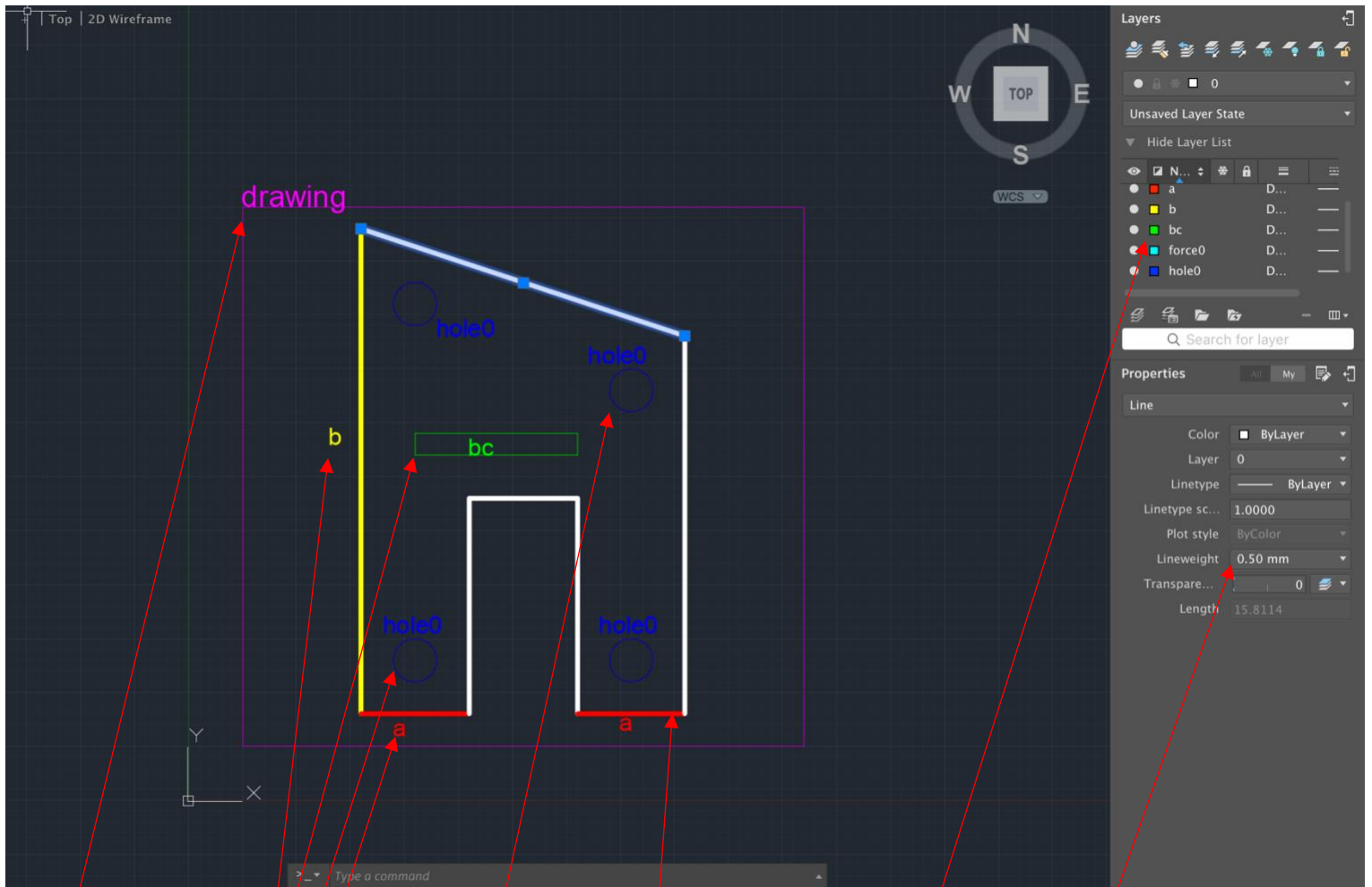
To define the frame within which the geometry is defined, create a new layer titled **“svgframe”**. Under this new frame, draw a rectangle (**LWPOLYLINE**) enclosing the geometry object. For best results, the rectangle should be as close to a square as possible. Then, use the **TEXT** tool (note: not **MTEXT**) and anchor a new text line that reads **“drawing”** to the upper-left vertex of the frame rectangle.

- **Defining the Geometry Units**

AutoCAD is natively unitless. To specify “units” that the geometry is in, enter **UNITS** in the AutoCAD command line and toggle the **Insertion Scale Units** to the specified units.

- **Need Help?** For additional help, please enter ‘help –dxf’ into the GUI command prompt.

- **Example DXF File (screenshot)**



Geometry  
frame

Variables

Inner  
geometry

Outer  
geometry

Variables  
(Layers)

Lineweight

## Test Sample Results

Test examples have been included with the program software package and utilize the two corresponding test DXF files. Below, the test sample text output result from using template **acadtest6.json**:

```
----- MODEL INPUT -----

// Material Properties //

Modulus of Elasticity: 20.8 GPa
Poisson's Ratio: 0.2
Plate Thickness: 0.15 m
Plate Dimension 'hole0': 2.25 m

// Mesh Properties //

Mesh Parameters [elType, dofsPerNode, elSizeFactor]:

[2, 2, 0.375]

Boundary Condition Parameters [[marker(s)], [values (m)]]:

[['b'], [0.0]]

Applied Load Parameters [[marker(s)], [values (N/m)], [angles]]:

[['hole0'], [2000000.0], [145.0]]

----- MODEL OUTPUT -----

Max Displacement [[element, vertex], disp] (m):

[[460, 0], 0.6012542424683786]

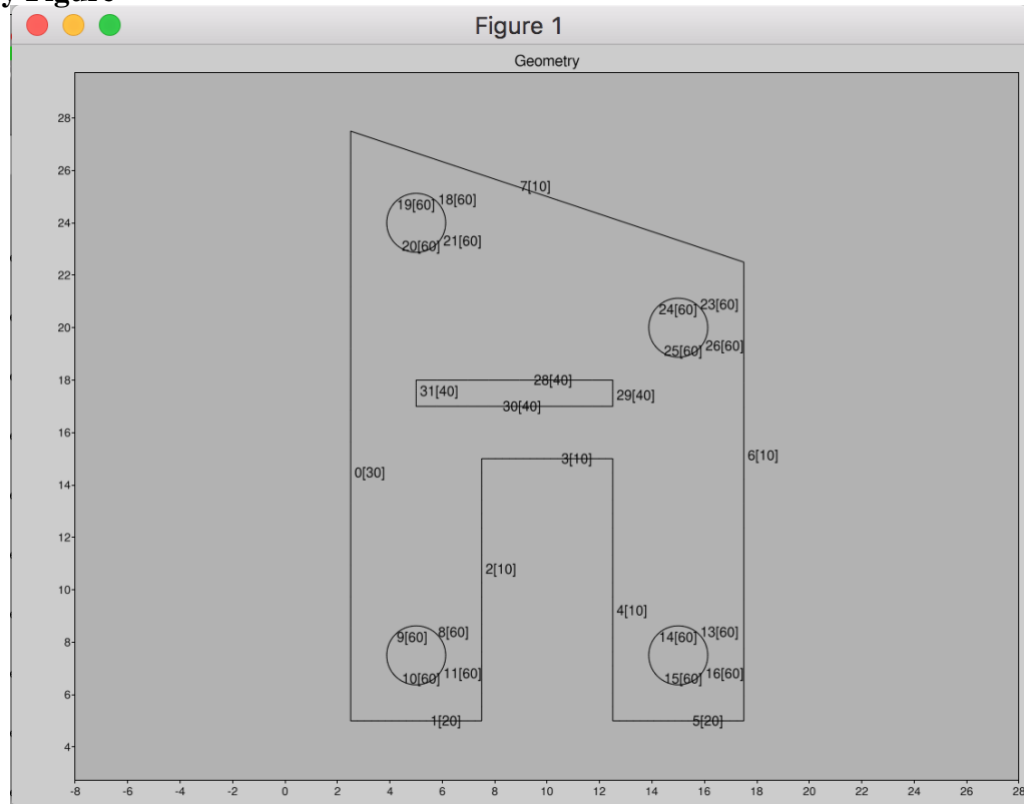
Max Element Von Mises Stress [Element, stress] (Pa):

[2650, 595857345.381724]
```

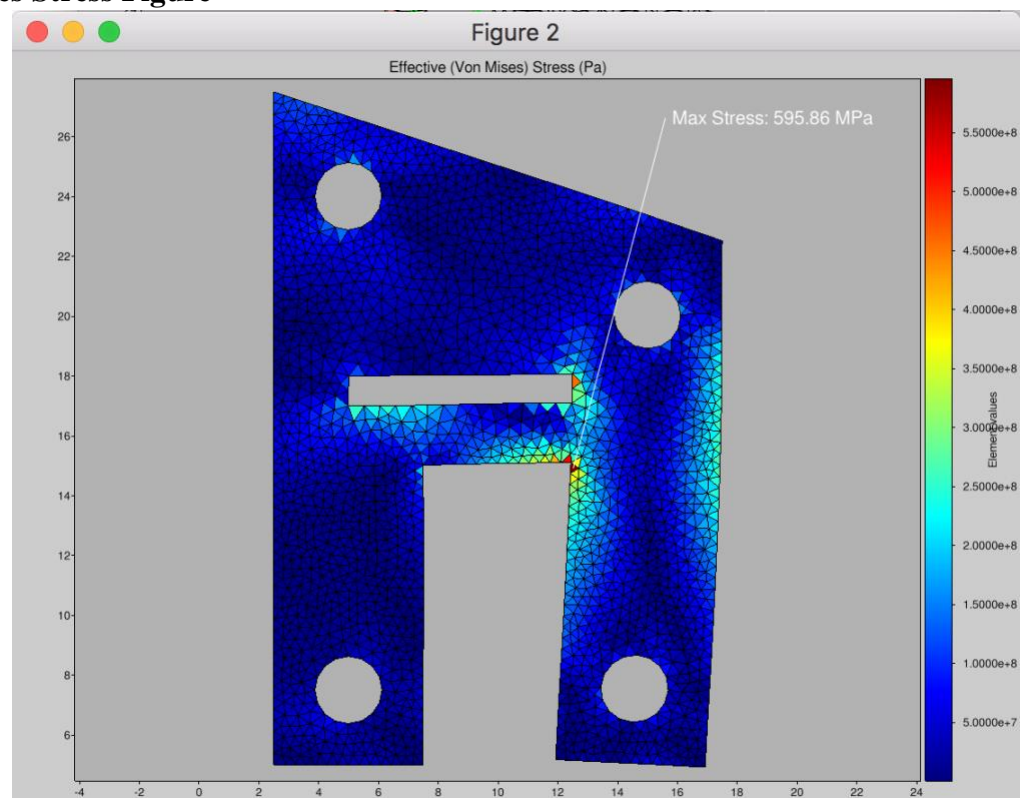


Below, the corresponding graphical presentation of the FEM output of the same example is shown below (all figures were toggled ON):

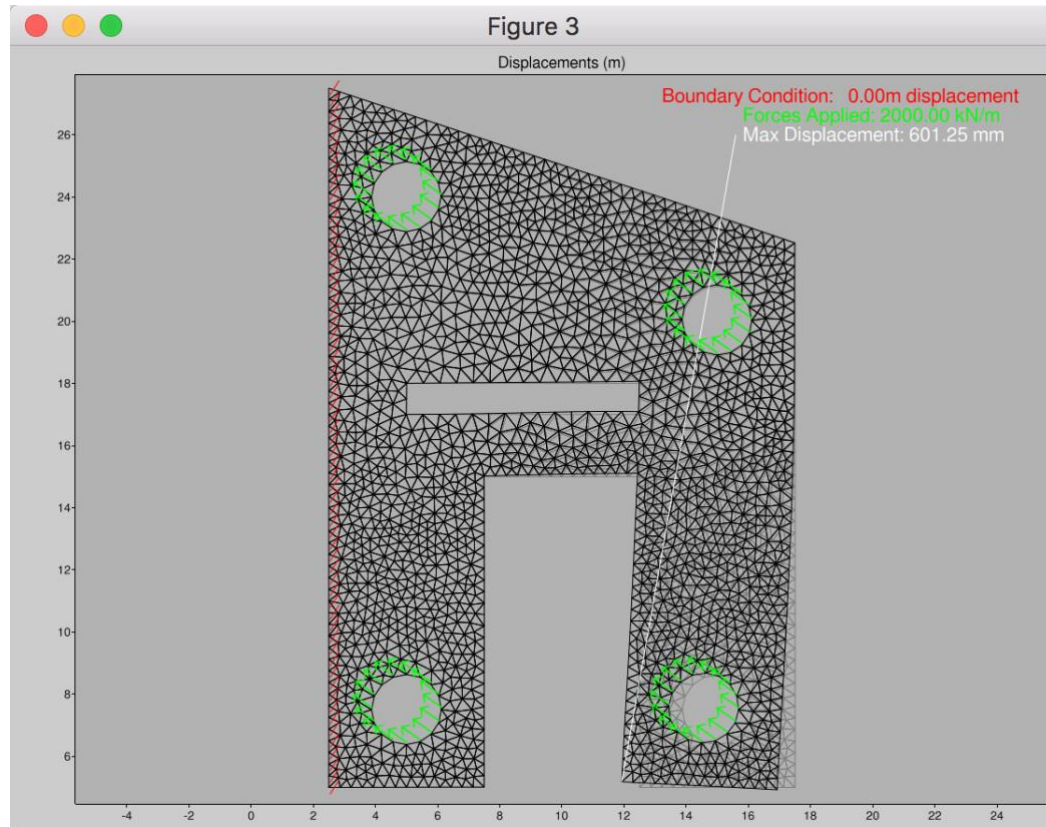
**Geometry Figure**



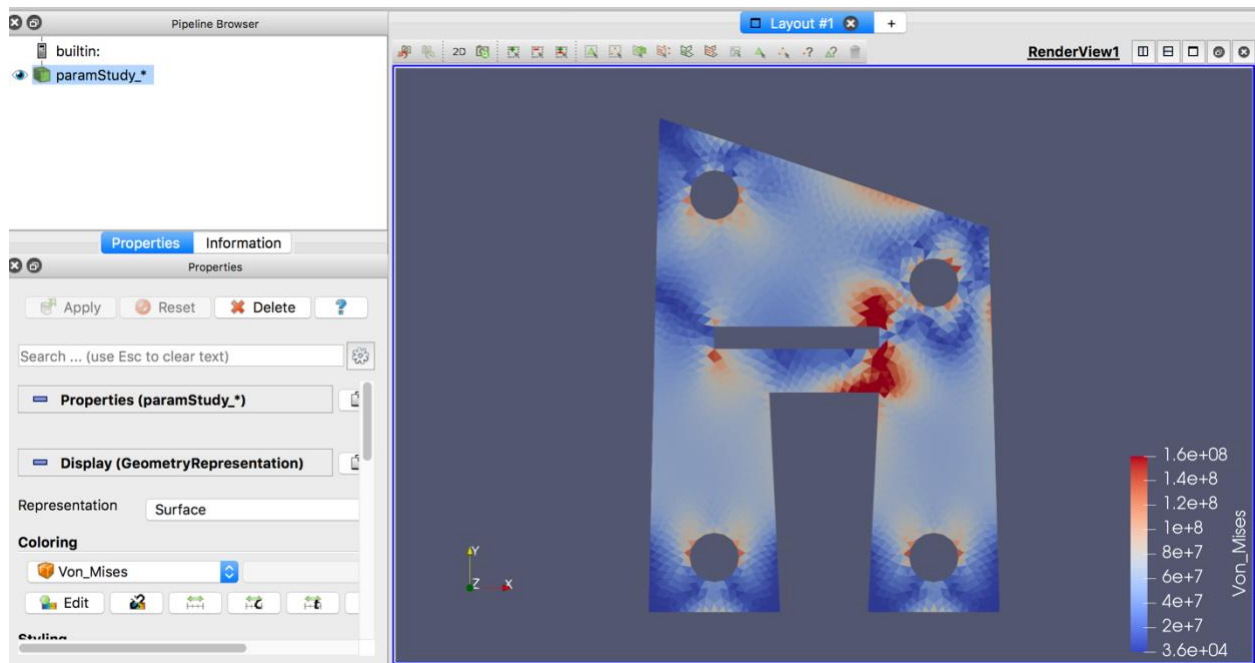
**Von Mises Stress Figure**



## Displacement Figure



Additionally, the output of a parameter study varying dimension **a** between 4.0 and 6.0 and dimension **hole0** between 0.5 and 2.25 diameter is included, with a screenshot of a .vtk cluster loaded in ParaView shown below:



## Source Code

All of the relevant source code can be found on the git repo:

[https://github.com/cremebrule/dxf\\_fem.git](https://github.com/cremebrule/dxf_fem.git)

Installation and executing instructions can be found on the git repo's README.md file.