About:

This program provides a way to quickly simulate the conductive heat transfer found in welding and additive manufacturing. The outputs it can produce include temperature and both solidification gradient and velocity, G and V.

For transient thermal conduction in welding and additive manufacturing, assuming uniform and constant thermophysical properties and neglecting the effects of fluid convection and latent heat, energy conservation can be written as:

|  |
| --- |
|  |

where is density, specific heat, temperature, time, thermal conductivity, and a volumetric heat source. Assuming a semi-infinite domain, Equation 1 may be solved analytically. The solution here for temperature at time t using a volumetric Gaussian heat source is:

|  |
| --- |
|  |

Where is the vector and is the location of the heat source.

The numerical integration scheme used is an adaptive Gaussian quadrature method based on the following nondimensionalization:

This program has 3 modes which each have their own uses. Mode 1 always tracks the temperature for all points which is useful when extracting the temperature data for a surface. Mode 2 only tracks the temperature for the melt pool. This mode is much quicker and used when only G and V are desired. Mode 3 takes a “snapshot” of the temperature field at the end of the final path segment.

The program requires 4 types of inputs: constants for the material in question (no temperature dependence), processing conditions (beam power, beam shape, preheat temperature), simulation parameters (mode, domain considered, number of points, timestep, and output frequency), and a path file which can contain both point melts and line melts.

Inputs:

**Material Information:**

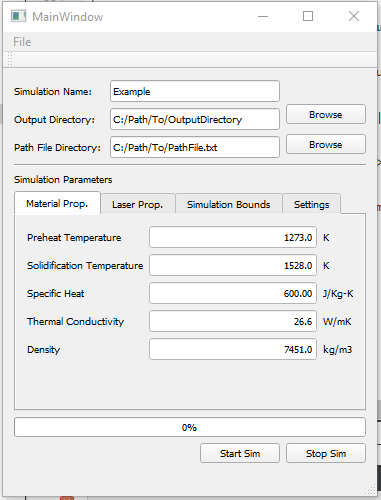
Solidification Temperature in units of

This is the temperature where the solidification gradients and velocities are calculated

Specific Heat in units of

Thermal Conductivity in units of

Density in units of



**Process Information:**

Width of the beam in the x, y, and z directions with units of

Typically, the x and y width are the same

The z direction represents the penetration depth of the beam

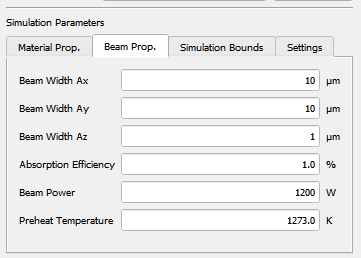
Usually very small unless keyholing occurs

Power of the beam in units of

Absorption efficiency of the beam

~0.3 for lasers and ~0.8 for e-beam

Preheat Temperature for the build in units of



**Simulation Parameters:**

Number of threads used for parallel computing

Most computers have at least 4 available

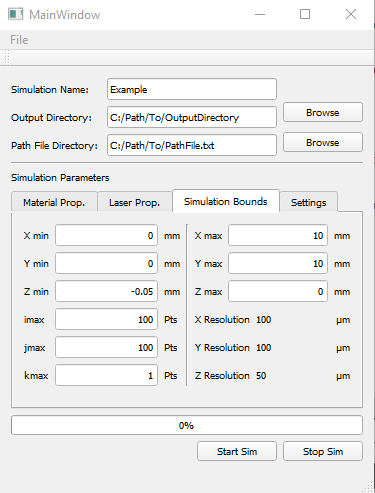
xmin, xmax – minimum and maximum x-coordinates for the domain considered (millimeters)

ymin, ymax – minimum and maximum y-coordinates for the domain considered (millimeters)

zmin, zmax – minimum and maximum z-coordinates for the domain considered (millimeters)

imax, jmax, kmax – number of points in the x, y, and z directions

[The maximum value is used when the number of points in a direction is 1]



**Simulation Settings:**

Number of Threads

The number of allowable threads the simulation will use on a computer

Mode

1 – Calculate the Temperature for all points at all timesteps

2 – Calculate the Temperature only for points in the melt pool

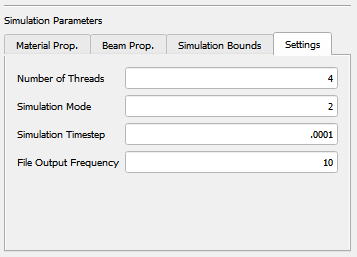
[Much faster for G and V only results]

Timestep in units of

[This is how often the temperature is calculated NOT the integration segment size]

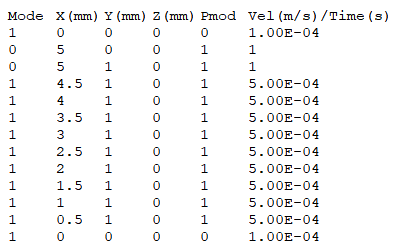
Output frequency

How many timesteps between output files



Path Files:

A path file will be a tab or space delimited file like in the picture below.



Mode indicates the type of melt occurring.

Mode 0 – Line Melt

Mode 1 – Spot Melt

X,Y,Z are the coordinates of the melt

Pmod is an additional power modulator

0 – Beam is off

1 – Beam is on

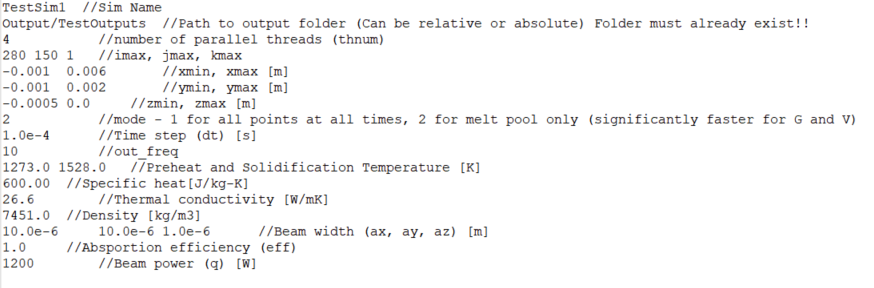
x – Beam is on at x times the power (allows for variable power)

Vel/Time is either the velocity of the line melt or the Time of the spot melt

NOTE: This is the velocity TO the point from the previous point! In the example above, the beam is at the origin and off for 0.1ms. It then travels 5mm in the x-direction at 1 m/s.

Running a Test Case

Every Simulation run from the GUI interface outputs a simulation file for the parameters used. This simulation file can be reimported into the GUI to rerun simulations with different path files, or simply for documentation. The parameter file follows the format below.

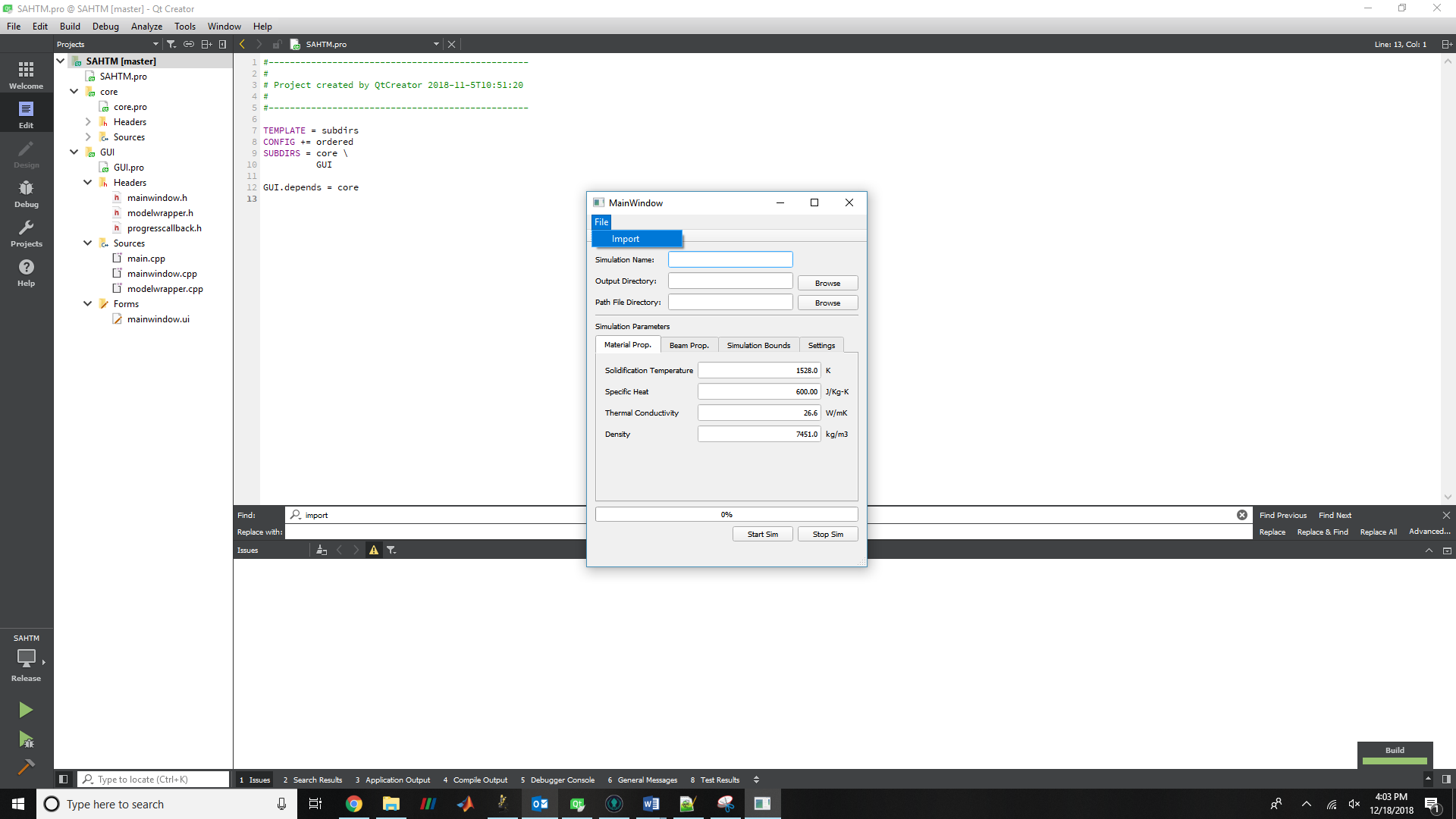
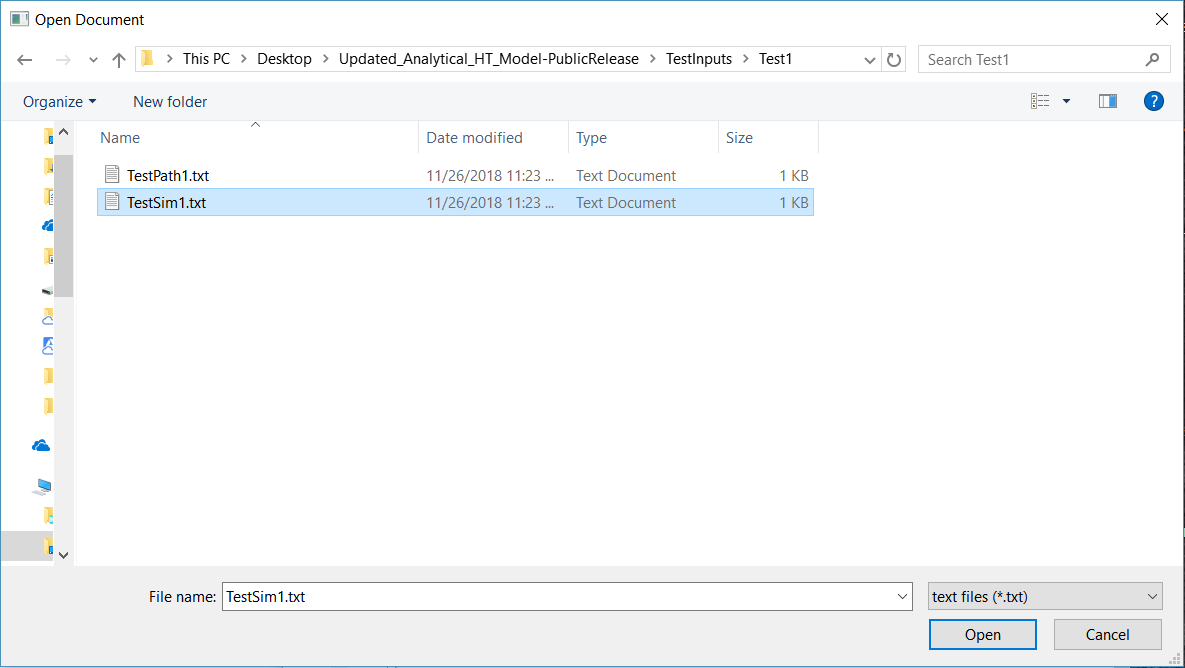


Some of these parameters have different units than the GUI. This is simply for the ease of input in the GUI interface. When reimporting, these will be converted to the units shown on the right side of the line edits.

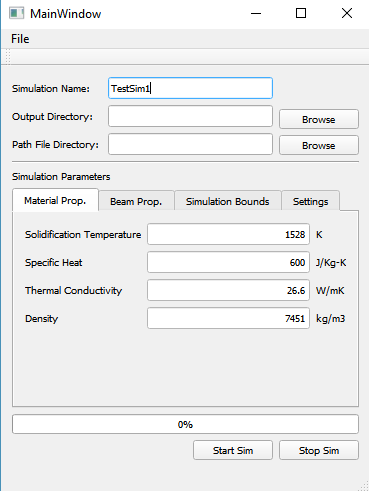
Another thing to know about the GUI, if importing simulations, the x min, xmax, ymin, ymax, z min, and zmax will be set to the values in the simulation file, but if a path file is then selected, the values will be overwritten to the maximum and minimum values in the path file.

So, to run a Test case, the following steps need to be done.

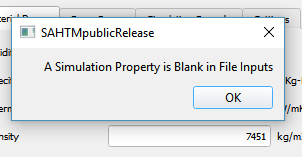
First, import the desired Test simulation.



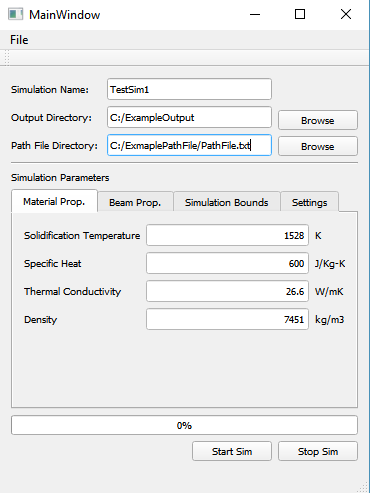
This will populate all the simulation parameters of the GUI with the desired input



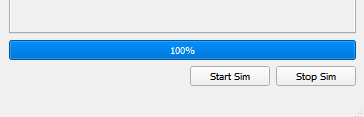
Notice, the output directory and the path file directory are not filed out. If you tried to start the simulation now, and error would appear for a blank field.



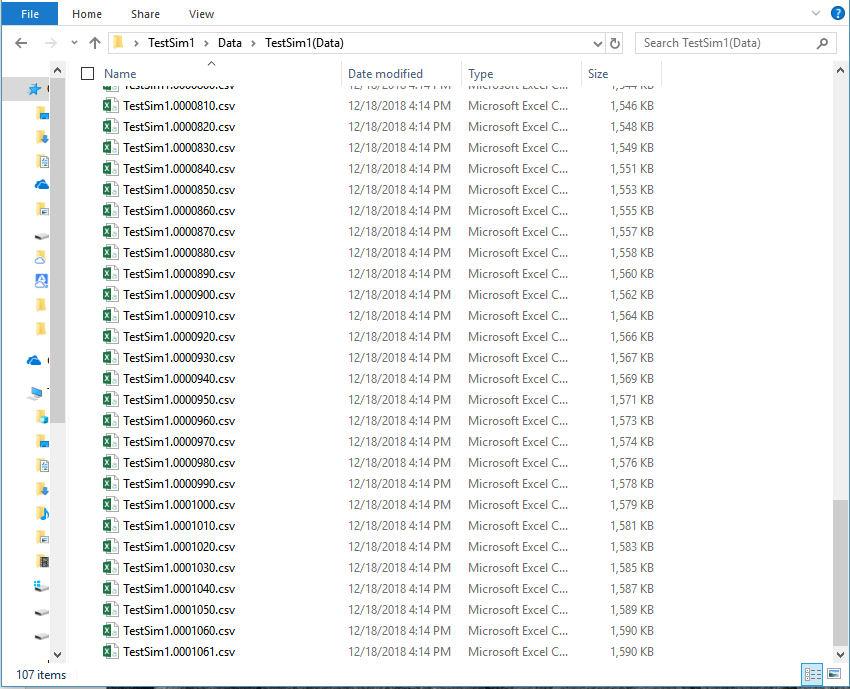
These remaining fields can be filled out using the browse button. The top browse button lets a folder to be selected for output location of the simulation. The second browse button allows the path file location to be selected. Once these are filled out, the simulation can be started by pressing the start button as shown below.

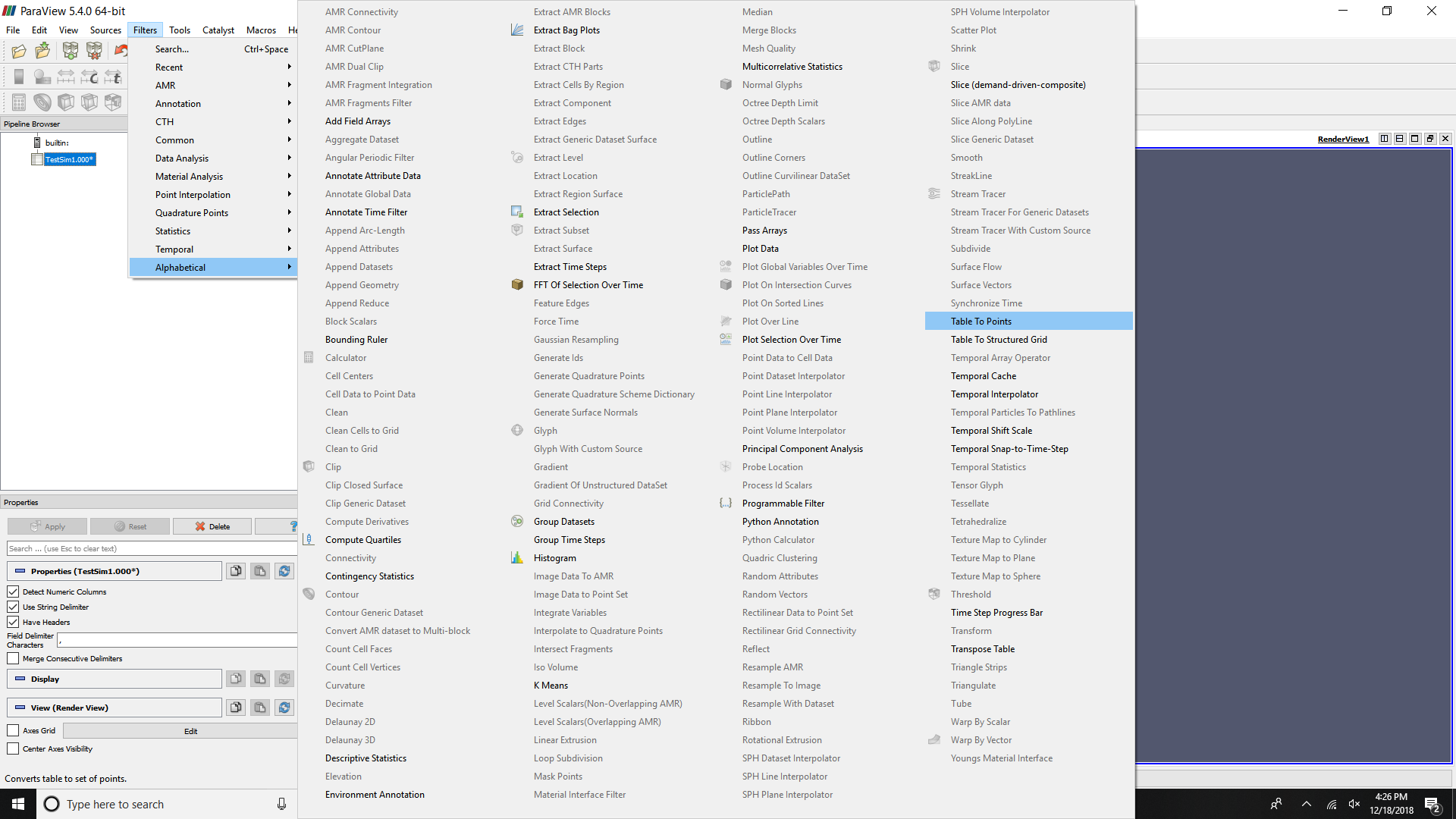


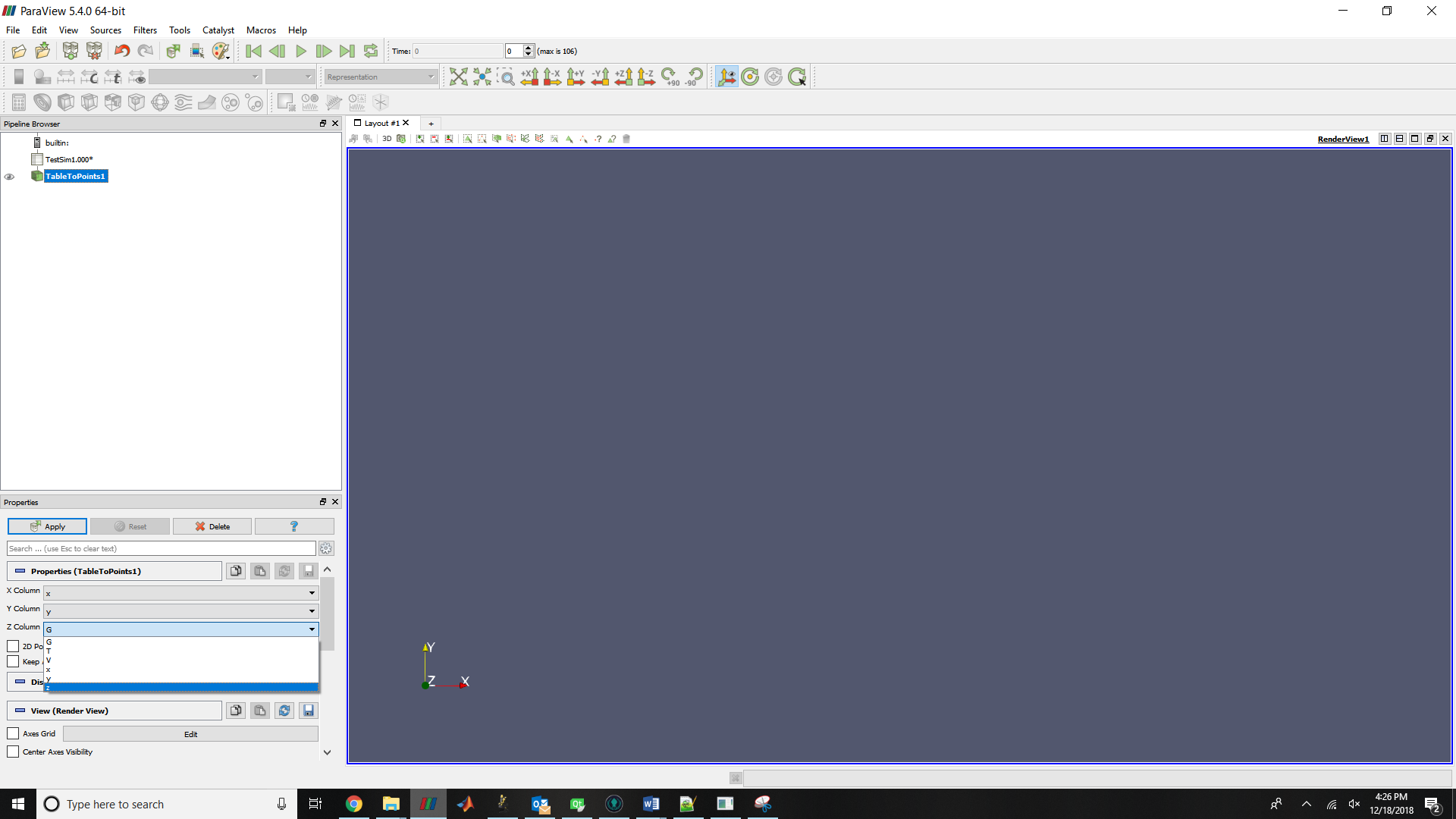
The simulation will run until stopped or exited by the user, or the simulation is complete indicated by the progress bar above the start and stop buttons. This bar may appear to hover around 99%, but it is just indicating that the path file is complete, and the simulation is calculating the final solidification values.



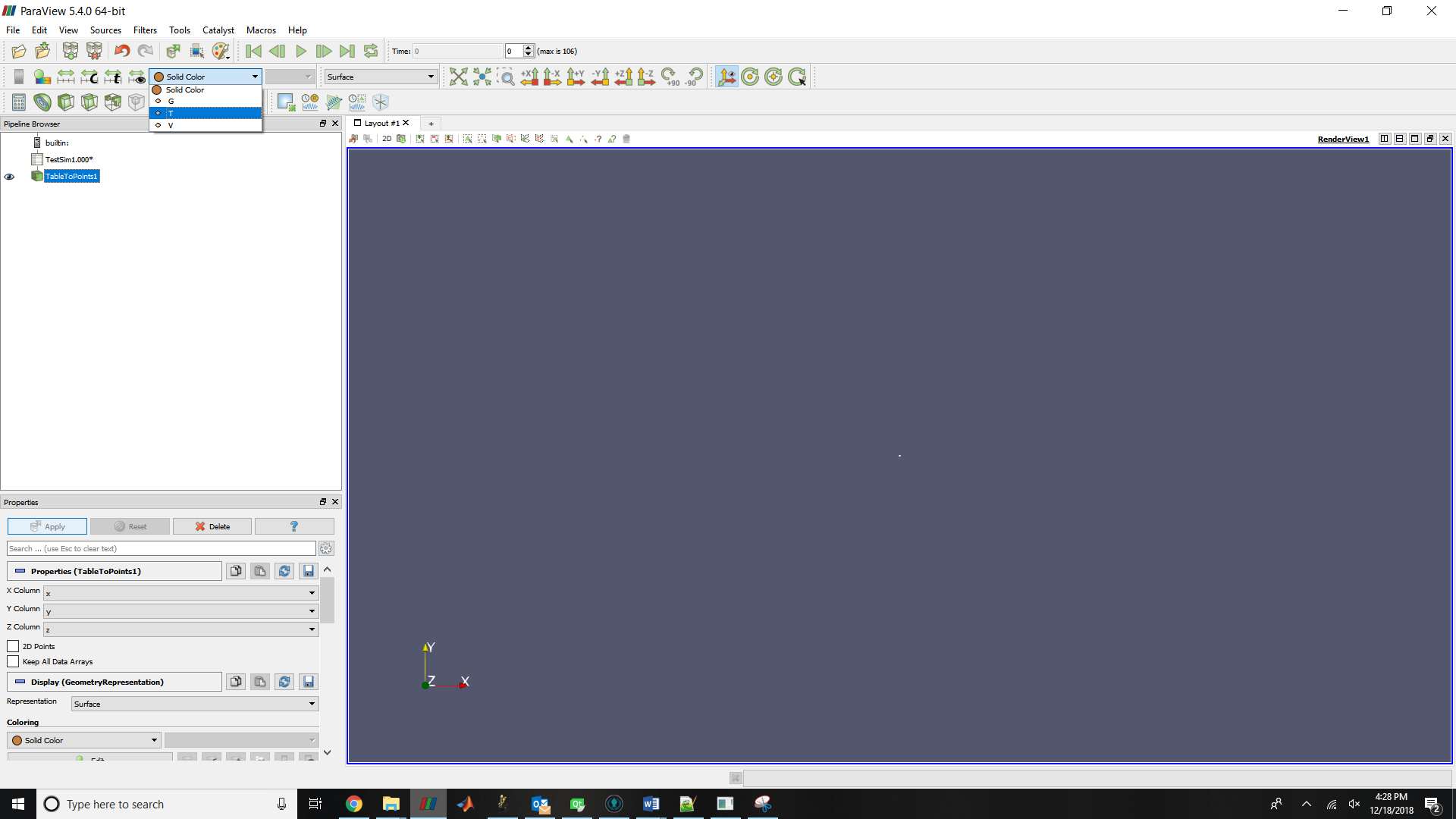
The simulation outputs .csv files at each desired timestep as seen below.



The GUI interface does not provide a method of viewing the data. A common program to use to visualize the CSV data is the popular open source software, ParaView. To view the data in ParaView, navigate to the output folder and import the CSV stack. Then, select the table to points filter, and set the X,Y,Z column headers in the properties panel and hit apply. 



Ensure that the view is turned on and set the settings to view the desired field.



The Data can be played as a movie and using ParaView can be exported to video. An example of a timestep can be seen below.

