Additive Manufacturing Part Level Distortion Sensitivity Analysis within Abaqus on a Thin-walled, Tubular Structure

Richard A. Deering

The Kansas City National Security Campus is operated by Honeywell Federal Manufacturing & Technologies, LLC for the United States Department of Energy under Contract No. DE-NA0002839

***Abstract**: As additive manufacturing (AM) evolves to become a more viable production solution in terms of cost, quality, and time, the need for predictive simulation of the process grows as well. After testing several commercial offerings to see how well they could predict deformation of various parts, Abaqus was found to be the most promising option and chosen for a more in depth analysis. The scope of this particular project was to examine the effects of certain simulation choices – from basics (mesh, time stepping, element type) to unique AM convergence techniques (full/partial activation, expansion time constant, follow deformation, etc.). Hundreds of simulations were run in Abaqus with various permutations and the resulting response on the final deformation and stress state was tracked. The results will be presented with charts and images to showcase the patterns (or lack thereof) produced by isolating each of these modeling choices on a thin-walled, tubular structure. The findings and conclusions are of value to anyone using Abaqus to simulate part-level distortion due to AM: for some – the study can provide a model or launching point for other in-depth sensitivity studies, for others – confidence in established procedures, or others still – insight to the myriad of options available with Abaqus's AM capabilities.*

***Keywords**: Coupled Analysis, Heat Transfer, Postprocessing, Residual Stress, Scripting, Thermal Stress, Tube, Visualization, Additive Manufactuing, AM, Thin-Wall, Distortion, DOE, Design of Experiments, Abaqus/CAE, Element Activation, Expansion Time Constant*

1. Introduction

Additive manufacturing (AM) is an evolving technology for fabricating products. The term AM is a broad envelope encompassing many specific processes each centered on the idea of building a part from the bottom up using a series of layers. All of these methods eliminate many of the constraints of traditional manufacturing methods by allowing for nearly limitless design opportunities. With additive, one can print entire assemblies rather than dozens of piece parts, manufacture organic, complex shapes, or manipulate build parameters to create gradient strength parts.

The specific process of interest for this paper is powder bed fusion (PBF). PBF is a common industrial process used to additively manufacture metal parts. The process entails

spreading a layer of metal powder over a bed. Then, a laser (or some other energy source) selectively melts regions of the powder that comprise the part. The regions are essentially cross-sections of the part, created and fed into the machine such that there is a cross-section for every powder layer within the build. This spreading and melting procedure is then repeated until all of the cross-sections of the part are printed. The end result is a fully dense, bulk part.

A common issue inherent to PBF is that thermal gradients throughout the course of the build begin to induce significant residual stress within the manufactured parts. At best, these stresses tend to warp the parts and can produce out of tolerance product. At worst, these stresses can fracture the part during the build and interfere with the recoater blade crashing the entire build. Typically, reducing excessive distortion requires manual trial and error consisting of repositioning the part, changing support structure strategies, or modifying machine parameter settings. This is a labor intensive approach which delays schedules and can cost thousands of dollars per build.

Avoidance of these failed builds could save a company both time and money while also accelerating the adoption of AM. As AM has gained more widespread traction, a dozen or so commercial simulation suites have begun to come out with specific offerings for AM. Each of these codes can fall into two main simulation categories – inherent strain method or transient thermo-mechanical method (TTMM). Both of these methods operate similarly by creating a mesh, partitioning the part into layers, then activating elements on a layer-by-layer basis and applying strain to the active layer. These strains are then resolved to predict the bulk distortion of the part. The main difference in the methods is how the applied strains are obtained. In inherent strain, the strain is predefined and exactly the same for each element making this method drastically faster, but potentially sacrificing accuracy. In TTMM, the applied strains are true thermal strains obtained by resolving the scan paths of the machine within a thermal model and feeding them into the mechanical model. These models are slower, but more closely mimic the physics.

Recently, Abaqus has released simulation capabilities to predict part-level distortion during additive manufacturing. They developed an entirely new framework to handle the many laser-mesh intersections. This framework allows the simulation to fully capture the provided scan path at whatever temporal or physical resolution is desired. It is a full transient thermo-mechanical model with a sequential, one-way coupling. First, a pure heat transfer simulation is run. This resolves the temperature history of the part which is then fed into a mechanical model as a predefined field. Then, the mechanical model runs to calculate (and resolve) the thermal strain associated with the temperature states. The output is a prediction of both deformation and stress.

2. Purpose

The purpose of this study is to examine this new simulation methodology and document the effects of various simulation choices on the outputs of interest – maximum displacement, maximum stress, maximum temperature, and runtime. This is important to understand how sensitive the solution is and what precautions must be taken during setup to ensure accurate, meaningful results. The study will also highlight useful convergence techniques and which parameter sets are most robust and accurate.

The study will consist of running a full factorial DOE changing 6 different aspects that ***ARE NOT APPLICATION SPECIFIC***. The factors chosen are all more closely related to mathematical formulation and solution of the problem rather than constraints/boundary conditions dictated by the physical system. As such, factors important to accurately represent the real world - machine temperature, scan paths, cooling effects, material properties, etc. - while important, were calibrated once beforehand and then held constant for all of the runs within the study.

3. Model Setup

The geometry used for the study is essentially a thin-walled, constant cross-section tube with a bend near the top of the part. It consists of two flat vertical walls and a hemisphere on either end connecting the two. Then, near the top of the tube, the opening curves at approximately a 45° angle. The change adds a bit more complexity and demonstrates the maximum possible deviation of the wall without requiring supports. The baseplate (12.5mm thick, stainless steel) was explicitly modeled. All units were consistent with mmNS unit system. See figure 1 below.

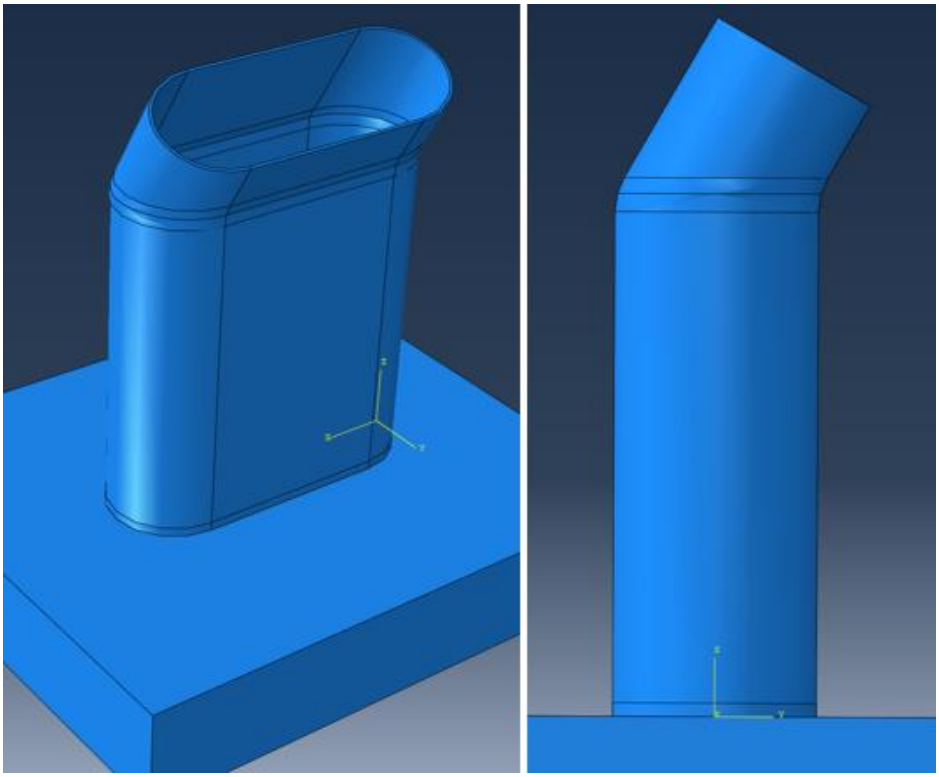

Figure 1. Thin-walled geometry

This geometry was chosen as a representative test piece because the most problematic parts to produce are thin walled structures. These structures are much less rigid and thus distort significantly more than bulkier components. Therefore, this class of thin-walled parts offers the best potential return on investment and is an ideal candidate for simulation. The size was chosen such that many of the parts could be built on a single build plate with the entire build taking less than a day. Many of these parts were manufactured and measured and so a plethora of data exists with which to compare the simulation predictions.

Physical parts were manufactured on a Renishaw AM250 machine as well as Concept Laser M-Lab using 316 stainless steel powder. The machine parameters were constant for all builds and were similar to preset strategies. While still attached to the build plate, these parts were blue-light scanned. This scanning procedure produced an .STL file which was then overlaid onto the nominal CAD geometry, through GOM Inspect (free), to visualize the deformation due to printing. See figure 2 below.

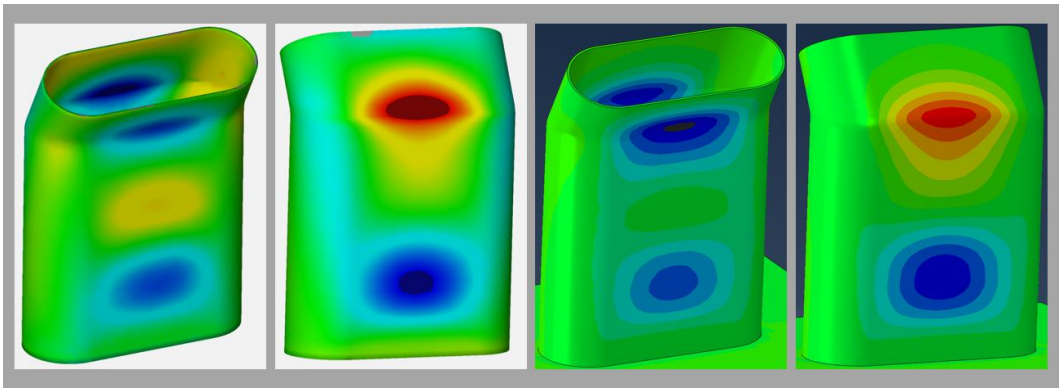

Figure 2. Baseline model comparison (equivalent scales [-0.4 mm to 0.4 mm])

Left: Measured front/back of physical part via blue light scan and GOM Inspect

Right: Predicted front/back displacement from Abaqus AM simulation

The base simulation model was created entirely within Abaqus/CAE (v2017.HF5) using the additive manufacturing plugin. The part was tied to the baseplate during both the heat transfer and static runs. This model was then run with defaults to validate some of our baseline boundary conditions. This serves to ensure the application specific assumptions which will be held constant for the study are adequate and predict a reasonable deformation profile and magnitude.

The following aspects are known to be important in simulation of AM, but were not varied in the study:

1. Absorption Coefficient – .68 based on the emissivity of stainless steel
2. Material properties – Temperature Dependent 316 Stainless Steel. See figure 3 below.

*Material, name=SS316

Units: self-consistent mmNs

*Conductivity	*Density		
14.89, 0.	8.03e-09,	*Expansion	*Specific Heat
14.89, 20.		1.46e-05, 10.	5.03e+08, 25.
15.79, 27.	*Elastic	1.46e-05, 20.	5.18e+08, 100.
16.61, 77.	195121., 0.27, 21.	1.48e-05, 27.	5.38e+08, 200.
18.28, 127.	189605., 0.27, 93.	1.52e-05, 77.	5.58e+08, 300.
19.77, 227.	186158., 0.27, 149.	1.56e-05, 127.	5.78e+08, 400.
21.21, 327.	182021., 0.27, 204.	1.63e-05, 227.	5.98e+08, 500.
22.59, 427.	178574., 0.27, 260.	1.69e-05, 327.	6.18e+08, 600.
23.99, 527.	174437., 0.27, 315.	1.74e-05, 427.	6.38e+08, 700.
25.33, 627.	170989., 0.27, 371.	1.79e-05, 527.	6.58e+08, 800.
26.58, 727.	166163., 0.27, 426.	1.83e-05, 627.	6.78e+08, 900.
27.81, 827.	162026., 0.27, 482.	1.87e-05, 727.	6.98e+08,1000.
29.18, 927.	157200., 0.27, 537.	1.9e-05, 827.	7.18e+08,1100.
30.34,1027.	151684., 0.27, 593.	1.93e-05, 927.	7.38e+08,1200.
31.55,1127.	146168., 0.27, 649.	1.95e-05,1027.	7.58e+08,1300.
32.7,1227.	139963., 0.27, 704.	1.96e-05,1127.	7.78e+08,1400.
32.5,1327.	132379., 0.27, 760.	1.98e-05,1227.	7.88e+08,1450.
32.5,1400.	124795., 0.27, 815.		

Figure 3. Abaqus Material Model

3. Scan Strategy – Scan Path and Screed Path, see below
4. Initial Temperature – 200°C based on early runs, ***calibrated value***
5. Build Temperature – 26°C room temperature build, no preheat
6. Cooling during build
    a. Convection - .018 mW/mm-°C to temperature of 26°C
    b. Radiation - .25 emissivity to temperature of 26°C
    c. Both values were taken from Abaqus's suggested parameters, no calibration

Most of the above conditions are easy to come by from either the physical build setup or standard material properties. The only additional boundary condition is applied to the bottom face of the build plate. In the heat transfer run, this face has a prescribed temperature of 26°C; in the static run, the face is constrained in all degrees of freedom.

One of the factors that has a large impact on displacement is the chosen "initial temperature" boundary condition. This is the reference temperature that thermal strain is calculated from. This value can have a large impact on both the magnitude of predicted displacement as well as the specific deformation profile. For these runs, a ballpark value was

chosen by running a simulation using a medium mesh (.2), medium timestep (25), C3D8I elements, and the default settings from the plugin (partial activation, follow def=NO, expansion time constant=0). The value was incremented by 25° C until the deformation seen was the same order of magnitude as observed in the physical part. Studies explicitly looking at this initial temperature with a couple different geometries have since been performed and a more true value seems to be ~300-375°C (cooling to 80°C build temperature). However the specific initial temperature value does not change the trends observed within this study. The value simply shifts the predicted displacement.

The scan path was not able to be obtained directly from the AM machine so a mock scan path was used. This mock path aimed to mimic the actual scan path as close as possible. It was generated through a series of scripts using Cura (free FDM software). The Cura framework was used with a modified "extruder" to replicate the same laser spot size and hatch spacing. The layer-by-layer rotation angle was also modified to more accurately capture the actual process. A portion of the path can be seen in figure 4.

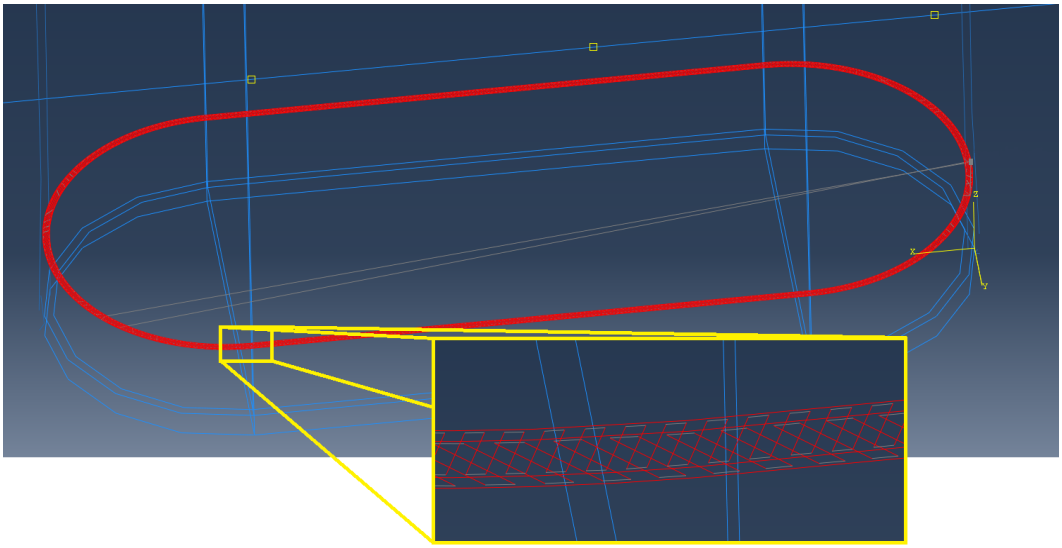

Figure 4. Scan path generation within Cura

4. DOE Design

Once the baseline runs were calibrated and the input decks worked as expected, the DOE runs were generated via python scripts to produce a full-factorial DOE with 6 different factors:

1. Follow deformation (2 levels)

    a. This determines whether or not the inactivated elements are allowed to move and follow the predicted deformation of the part
    b. Tested values – YES, NO
2. Mesh density (3 levels)
    a. Determines the dimensional resolution of the part
    b. Mesh was controlled by a global seed size. This seed size was picked such that the finest mesh had 3 elements through the wall thickness and the coarsest mesh had only 1
    c. Tested values - .1333, .2, .3 (1.5x factor between seed values)
3. Time step (4 levels)
    a. Determines the temporal resolution of the part, direct specification
    b. Time step was chosen so the finest resolution was approximately the time to build 1 physical build layer (~9sec/layer for the constant cross section, 8sec dwell)
    c. Tested values – 12.5, 25, 50, 100 (2x factor between each time step)
4. Expansion Time Constant (3 levels)
    a. Determines the amount of time over which induced thermal strains are applied
    b. Tested values – 0*time step, 2*time step, 5*time step
5. Activation Method (2 levels)
    a. Defines how elements get activated
    b. Tested values – Full, Partial
6. Element Type (3 levels)
    a. Hex (reduced integration and incompatible modes) vs. Tet
    b. Tested values – C3D8R, C3D8I, C3D4

Again, these factors were chosen due to the fact that they are not dictated by the physical builds or machine settings. Each of them are choices the simulation engineer must make for any AM job run within Abaqus. Understanding how these choices affect the outputs will allow the analyst to make appropriate choices and inform them of the risks of varying certain parameters.

The intent was for each of these runs to be solved on an HPC cluster using 8 CPUs. Due to runtime concerns, the fine mesh runs were instead run on 128 CPUs. All of the submittals were done in two job batches to ensure that the heat transfer fully completed before the static initiated. Bash scripts handled the creation of subfolders, running of jobs, monitoring of jobs, transfer of results, and deletion of excess files (especially the large heat transfer .odbs).

Analysis of output was done via python scripts. The scripts would open the static .odb file and record the status of the job (Failed/Completed… if failed then at what time it failed), the maximum stress and the maximum displacement at the final frame, the temperature at t=5150 (arbitrarily chosen time during the constant cross-section portion of the build). It would also take a series of snapshots of each of these quantities. Runtime was pulled from the .msg files. All of this data was then exported to a .csv file that could be viewed in Excel or Minitab.

5. Results

The following result plots were produced in Minitab using a subset of the data indicated in the DOE design. First, resource limitations prevented the running of every finest mesh size so all of those runs have been removed from the analysis. Additionally, the C3D8R runs were all removed due to the vast majority of the coarse mesh runs failing; these failures occurred early in the run and the corresponding stress and displacement values are drastically low. Finally, only runs with follow deformation=NO were used. Follow deformation=YES resulted in several additional failed runs, which is undesirable. Since follow deformation=NO produces less failed simulations it makes sense to isolate those runs. This is especially helpful considering that having both simply obfuscates the overall data by averaging follow deformation=YES's drastically higher values with the lower values of follow deformation=NO. See figure 5 below.

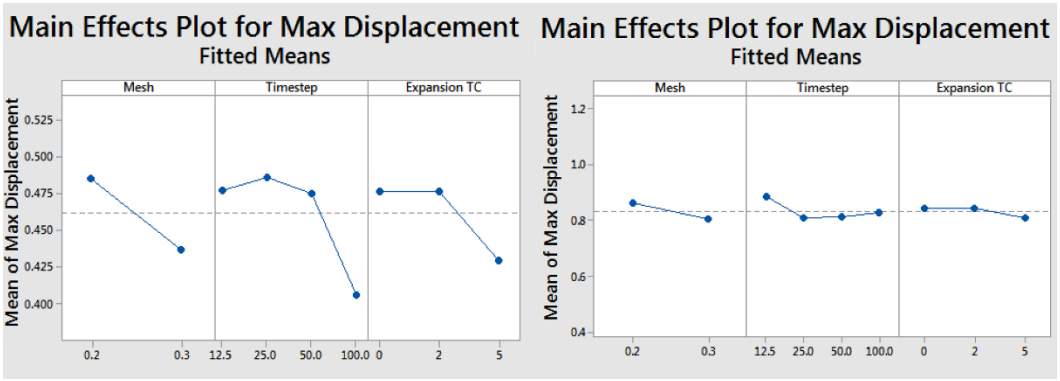

Figure 5. Effects for follow deformation=NO only (left) vs. Effects when also including follow deformation=YES results (right)

There are two different plots for each output variable of interest. First, the "main effects plot" which indicated the effects that each of the individual factors had on the output. The slope of the lines denotes the severity of the effect; the steeper the line, the larger an impact that factor had. The second plot is an "interaction plot". Within these plots, each section represents a certain combination of factors. The main indicator of an interaction between the two factors listed is non-parallel lines. Parallel lines indicate that the combination is insignificant whereas non-parallel lines mean that the specific combination has a significant impact on the result. All of the plots are included as is so that the reader can make their own conclusions. After the plots, noteworthy takeaways are compiled along with any observations that may explain the stated behavior.

5.1 Temperature

High-Impact Factors: Mesh size, time step, element type, and activation scheme
Non-Impact Factors: Expansion time constant
Range: 26.06-28.39°C

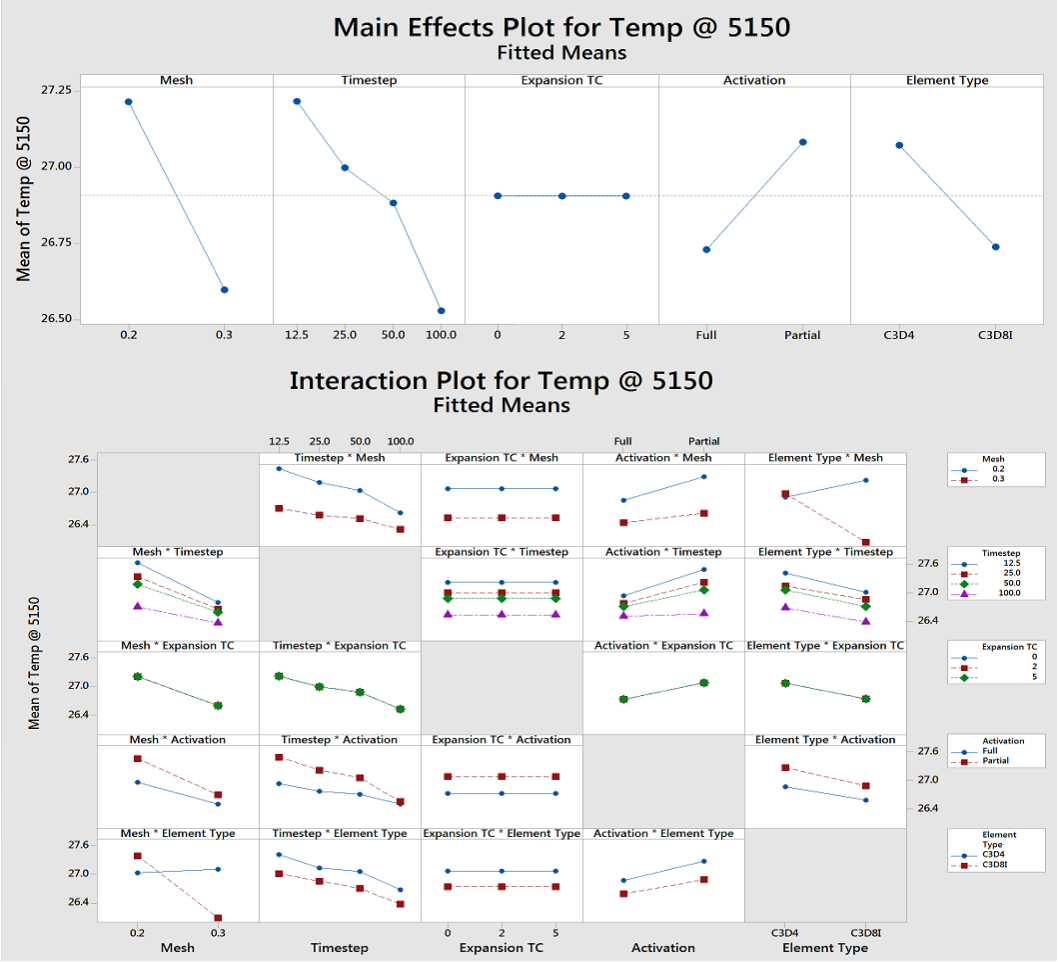

Figure 6. Minitab factorial plots for temperature

Temperature results were as expected. The main effects in order of impact were mesh size, time step, element type, and activation. Expansion time constant was the only factor that had no impact on temperature; which is to be expected as it is present in only the static analysis. Essentially as the simulation became more refined spatially and temporally, the temperatures rose. The smaller time steps and mesh sizes resulted in higher temperatures as did the smaller element type (tets) and sub-sectioning the element through partial activation. However, the total range of temperature values were 26.06-28.39ºC. All of these simulation choices had very minor effects on the magnitude; essentially in all cases, the activated elements were contracting from their initial temp (200ºC) to room temperature (26ºC) in a single time step. Partial activation appeared to be more sensitive to mesh size than full activation which was mostly driven by the fine mesh and finer time steps; at the coarse end of the spectrum both activation schemes produced similar

results. Additionally C3D8I elements were very sensitive to changes in mesh, whereas C3D4 elements were not. This is most likely due to the fact that with 2 elements through the thickness, there is now a node in the middle of the wall which takes longer to cool than both of the exterior nodes present with only 1 element through the thickness. With a tet mesh, there already exists some interior nodes even for the largest mesh seed.

5.2 Maximum Stress

High-Impact Factors: Activation scheme, element type, and mesh density
Non-Impact Factors: Expansion time constant
Range: 956- 2815MPa

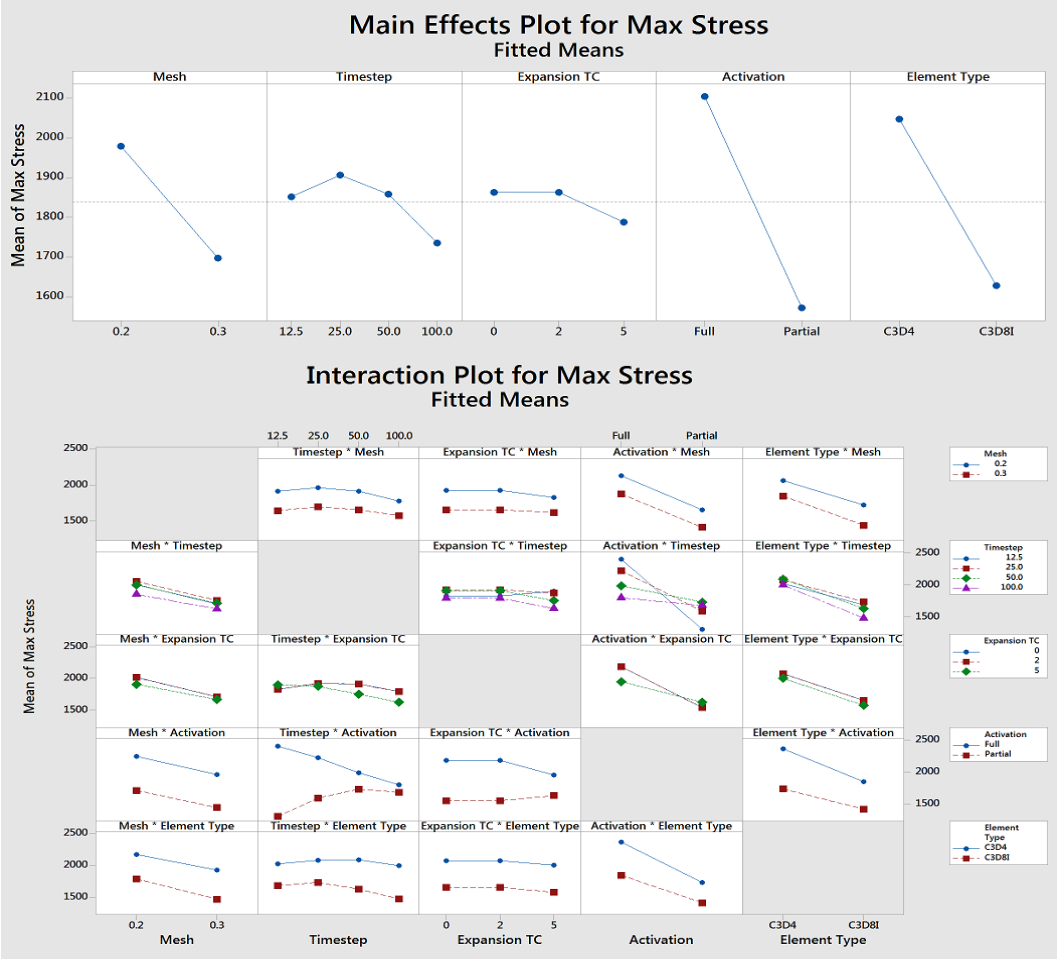

Figure 7. Minitab factorial plots for stress

With the temperatures from the heat transfer run not varying drastically, one would expect to see similar results for both stress and displacement as the driving force for the static analysis is the output of the heat transfer. However, even with such seemingly tiny changes, the predicted maximum stress varied from 956- 2815MPa. The main effects in order of impact were activation scheme, element type, and mesh density. Expansion time constant had no effect when the value was 0 or 2, but showed a drastic decrease at 5. This would suggest that 5 is too great a value which was especially apparent for the higher time steps. Additionally, activation schemes had a drastic effect on both time step and element type. When choosing full activation, there is more uncertainty between the results of the tet and hex mesh than the partial activation runs. As for time step, the activation scheme chosen dictates the trend of the stress values as the time step becomes finer. The stress increases using full activation and decreases when using partial activation. It is uncertain why this is and could be the reason why overall time step does not have a clear trend in terms of maximum stress predictions.

5.3 Maximum Displacement

High-Impact Factors: Activation scheme, element type, and mesh density
Non-Impact Factors: Expansion time constant
Range: .22-.66mm

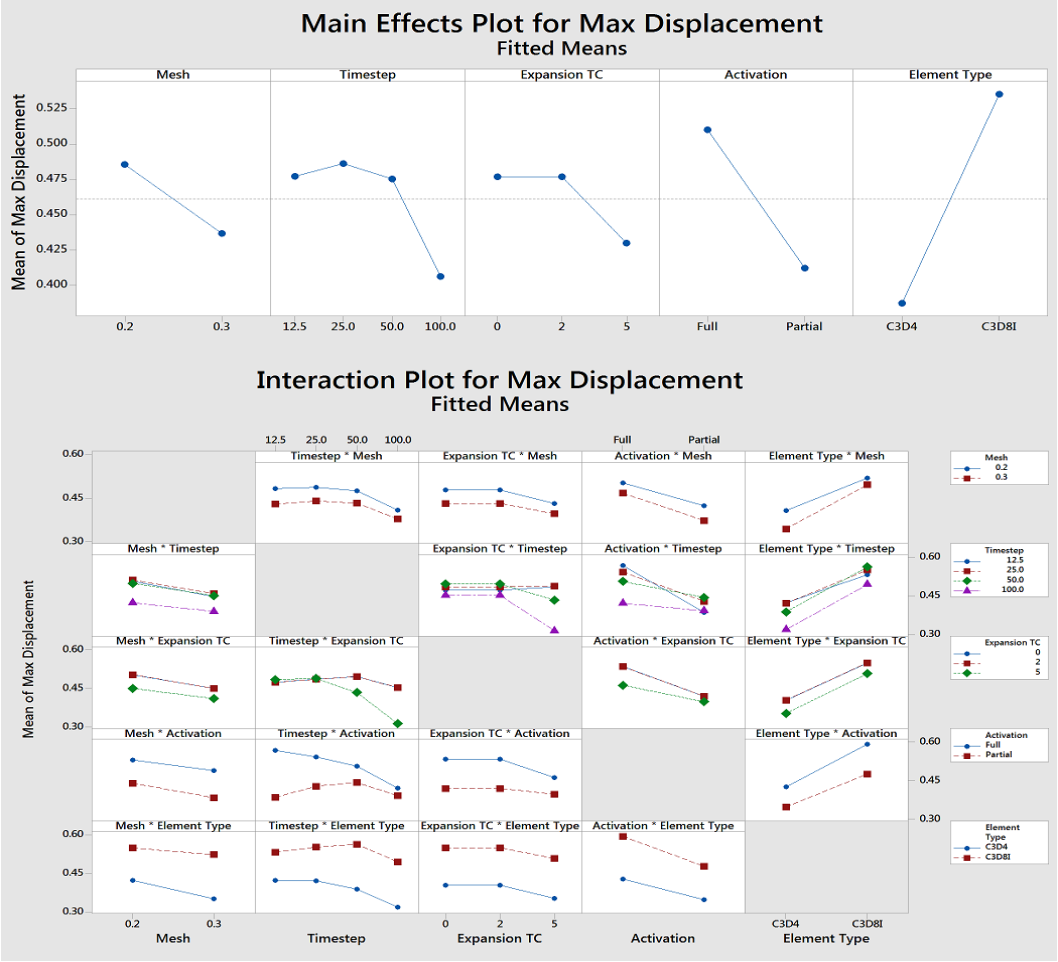

Figure 8. Minitab factorial plots for displacement

Maximum displacement follow all of the same overall patterns as stress, which is to be expected. The main effects, again, in order of impact, were activation scheme, element type, and mesh density. The individual values ranged between .22-.66mm. Expansion time constant had no impact when the value was 0 or 2, but showed an even more drastic decrease at 5 which further cements the higher values as being undesirable. In addition, the 100 second time stepping showed notably lower values for both meshes which may suggest the temporal resolution is too coarse. The activation scheme chosen dictates the trend of the displacement values as the time step becomes finer. The distortion increases using full activation and decreases when using partial activation which mimics the same behavior shown in the stress. Also, choosing full activation exacerbates the impact of higher value expansion time coefficients. Hex mesh appears to be slightly more resilient to changes in time steps.

5.4 Runtime

High-Impact Factors: Time stepping, mesh, and element type
Non-Impact Factors: Expansion time constant, activation scheme
Range: 21min – 35hrs

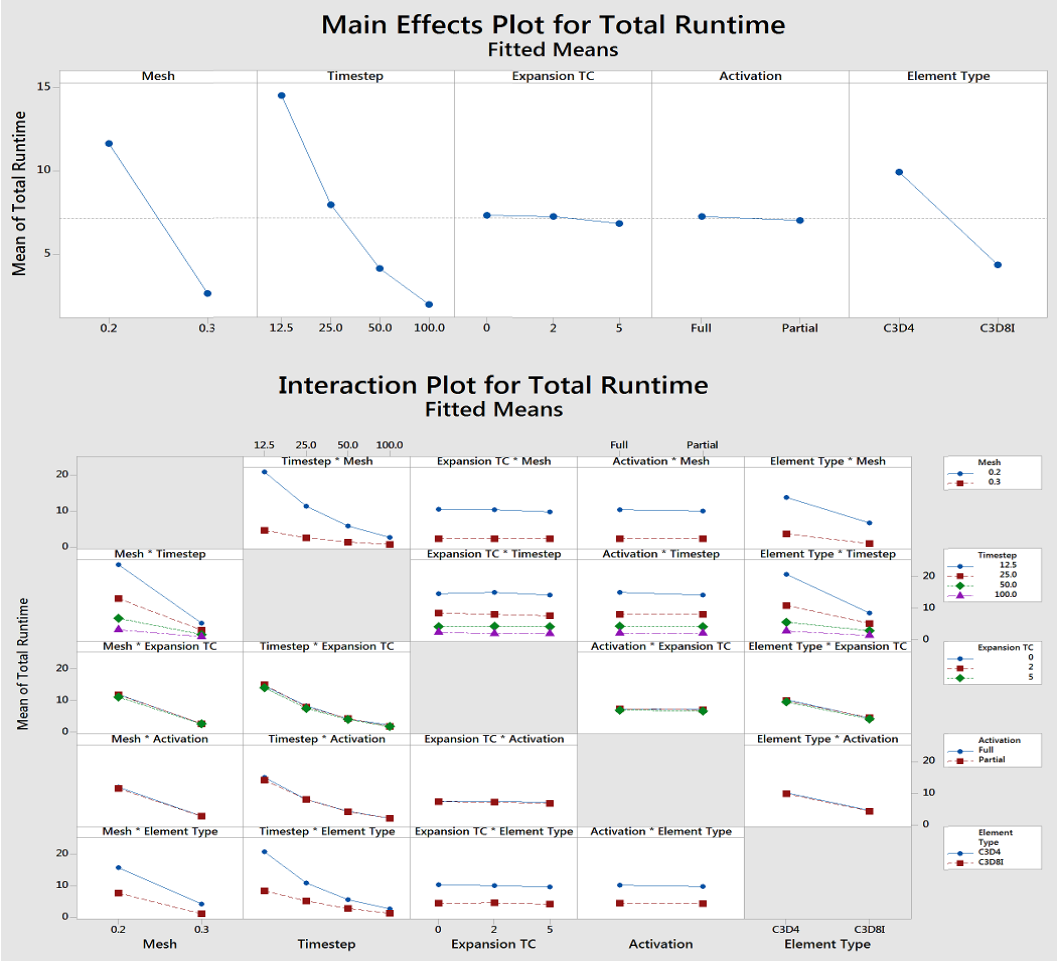

Figure 9. Minitab factorial plots for runtime

Most of the runtime results are rather trivial. Runtime is dictated primarily by time stepping, mesh, and element type. This is exactly as one would expect. Each of the simulations used 8 CPUs and thus the more elements present either with a finer mesh or choosing tets over hexes results in a slower solution time. Likewise, the number of increments that must be solved

which is dictated by the time step will increase the runtime. The noteworthy insights were of the aspects that did not change the runtime. The activation scheme did not have an impact. Both partial and full activation averaged about the same runtime. The same can be said for expansion time constant. There was no notable gain in runtime by using a larger expansion time coefficient.

6. Conclusion

Overall, Abaqus is capable of replicating the same deformation profiles seen in physical builds of thin walled specimens. However, the simulation choices that an analyst makes do have a drastic impact on the overall prediction. The choice of allowing follow deformation in particular is crucial. Follow deformation=YES does improve convergence in some cases (such as when using coarsely meshed reduced integration elements), but usually allows far too much distortion, ultimately making the solution meaningless. It is also more costly in terms of runtime. Mesh density did not have as large of an effect as predicted, but the predicted displacement does increase almost uniformly as mesh density increases (for the 3 seed sizes indicated in the DOE). Time stepping had a non-linear impact on many of the results of interest including displacement. 100 seconds (~6 physical build layers) seemed to be too coarse and 12.5 seconds (~1 physical build layer) seemed needlessly fine. Somewhere between seemed adequate in terms of accuracy and runtime. It did not show a large dependence on mesh density either; the two are fairly independent. Element type is as important in AM as it is in other analysis types. For thin walled structures incompatible mode hexes were about 2x as expensive in terms of runtime, but always converged and did not necessitate as fine of a mesh. Activation scheme is dependent on other simulation choices; full and partial both have merits. Full proved to be more robust, especially with a tet mesh, but partial is a closer representation of the physics and doesn't sacrifice runtime. Expansion time constant did not seem to have an impact in any way until it got too large (~5x the time step).

Recommended default settings:

Follow deformation = NO

Mesh resolution: 1 element through thickness is reasonable

Element type = C3D8I

Activation Scheme = Partial

Time step = 2-4 physical build layers

Expansion time constant = 2*time step (recommended by Abaqus… virtually identical to 0)