Example Nonparametric Optimization Cases for Additive Manufacturing Using Tosca and Abaqus

Juan F. Betts, Mark A. Walker, Arash Alizadeh
Front End Analytics LLC

Abstract: two example nonparametric optimization cases for AM using TOSCA and ABAQUS are presented. For the first case, we optimized a disc brake caliper, where loading conditions were driven by the master cylinder, the contact areas of the disc rotor, and the envelope limitations within the wheel. The objective function, design constraints, and the targeted design response drove the outer topology of the caliper casting. TOSCA's topology optimization established the primary geometry, and subsequent validation of the parametric reconstruction was performed in ABAQUS. The resulting optimized geometry removed 62% the caliper bridge volume from areas of low stress and replaced it (as needed) in high stress areas. The resulting overall brake caliper weight reduction was nearly 10%. For the second case, we optimized the neck of a prosthetic hip implant, where the loading condition was an extreme side load applied to the implant's stem, and the envelope was set by the clearance requirements of the femoral head during joint articulation. Similar to the caliper case, TOSCA's topology optimization minimized the material volume of the neck in free space areas (where there were no other requirements such as assembly or mounting to neighboring components), while maintaining internal stresses well below the neck's yield strength. The resulting optimized geometry removed 48% of the neck's free space for a total component weight reduction of over 10%.

Keywords: topology optimization, non-parametric optimization, design optimization, automotive, ABAQUS, TOSCA, disk brake, caliper, analysis verification, analysis validation, minimum volume, minimum weight, Finite Element Analysis, implicit analysis, additive manufacturing, hip implant, orthopedics, life sciences

1. DISC BRAKE CALIPER OPTIMIZATION

A. INTRODUCTION

Automotive components are mass produced and the amount of material used in a given component is a significant cost driver. Reducing the amount of material used and scrap produced in the manufacturing process is of significant economic value in this market. In addition, reducing the amount of material leads to performance improvements by making the vehicle lighter.

In this case, a disk brake caliper was optimized specifically in its “bridge” region, which joins the two halves of the caliper together around the disc. The optimization was set to reduce the overall volume of the bridge section while not exceeding a Von Misses stress level of 250MPa, which was less than the yield stress of 270MPa for the bridge material (AA354 aluminum). This example discusses the process to build this optimization study and the results obtained.

B. INITIAL FINITE ELEMENT RESULTS

The topology of a disk brake caliper was optimized, specifically in its “bridge” region, which joins the two halves of the caliper together around the disc. The process involved stripping down the caliper assembly model to the targeted component in CAD (SolidWorks in this case), exporting to a neutral format (e.g. STEP), then importing the geometry and finishing the analysis model within ABAQUS CAE. Figure 1 shows the geometrical simplification in SolidWorks.

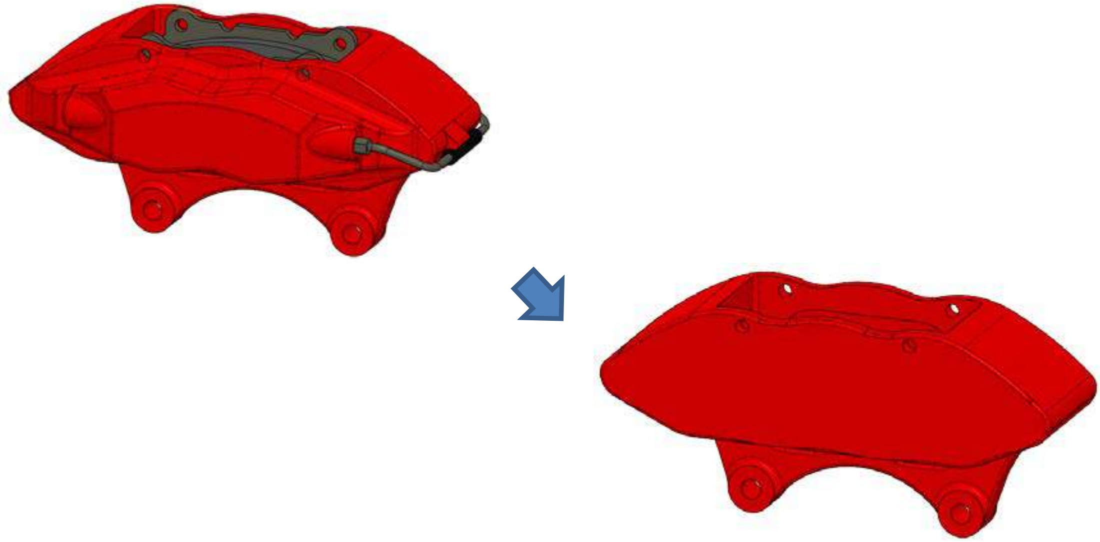


Figure 1: Geometric CAD simplification in SolidWorks

Once in ABAQUS CAE, caliper mounts were fixed within the analysis model and a typical maximum master cylinder pressure of 1000psi (6.9MPa) was set to operate on the inside walls of the caliper. A more complex model could have included the pistons passing through the caliper walls to compress pads against the disc, with the pressure over the piston surface area creating the inward clamping and outward reactionary forces. However, this simplified geometry and analysis model lets us focus on the comparative study of the bridge geometry. Figure 2 shows the ABAQUS CAE model imported from SOLIDWORKS.

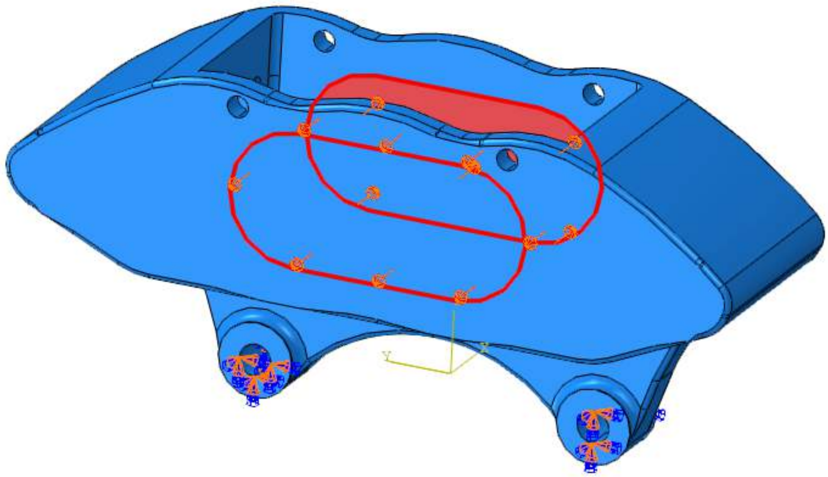


Figure 2: Analysis model built in Abaqus CAE, showing pinned mount locations and simplified pressure loads

Material properties were then applied to the model. The material used for this analysis was heat-treated AA354 aluminum casting alloy, with an approximate yield stress of 270MPa. Boundary conditions were then applied to the model with pinned locations and pressure surface as shown in Figure 2. A single general, static step was then created using a standard Ten-node tetrahedral element (C3D10) mesh, with an approximate global seed size of 3mm (this part is 85mm wide), and an implicit analysis was sent to be solved.

The initial analysis results showed that the maximum Von Misses stress levels were in the order of 219-255MPa in the underside of the targeted bridge geometry. Figure 3 shows the Von Misses stress rainbow plot for the caliper. Several areas of the bridge caliper show low stress level, indicating that material could be removed without affecting the structural integrity of the part.

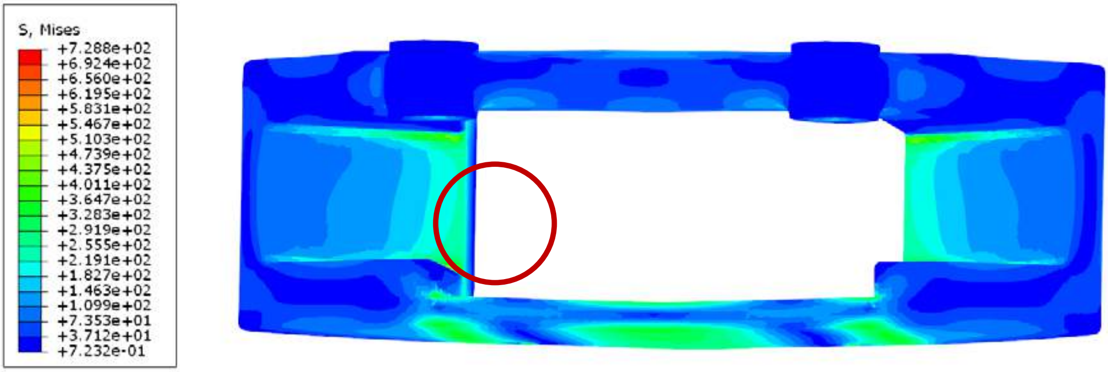


Figure 3: Analysis results of original model, showing stresses under bridges up to 255MPa

C. TOPOLOGY OPTIMIZATION

Keeping the same boundary conditions, a nonparametric topology optimization of the caliper bridge was performed using TOSCA. TOSCA is embedded into ABAQUS CAE allowing this procedure to be run directly from ABAQUS CAE. The targeted optimization design space was the caliper bridges. However, before setting up the optimization with TOSCA, the CAD model was altered to increase the volume of the bridges slightly to their maximum extent allowed by the system (inner diameter of the wheel, plus clearance). The CAD model was then imported into ABAQUS CAE and partitions were created to target the optimization region. Figure 4 shows the identified areas (in green) where the topology optimization would be performed.

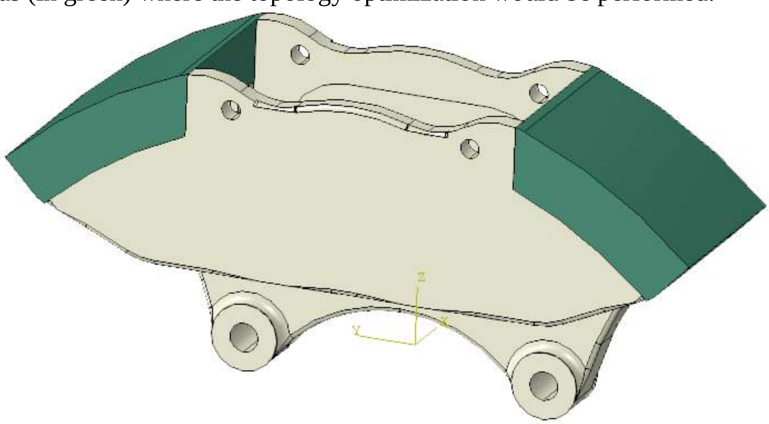


Figure 4: Identified regions of the analysis model (variable design space in green, fixed in tan)

The same analysis conditions were then set in the updated model, with the same material and mesh conditions as originally used as shown in Figure 5 and another FEA analysis was run.

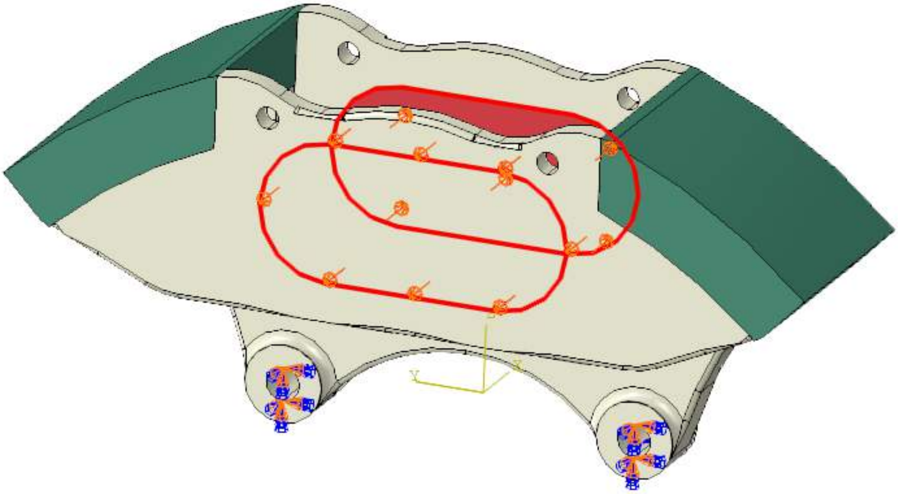


Figure 5: Loading of the optimization analysis model, showing the same pinned locations and pressure surfaces

Figure 6 shows the resulting Von Misses stress rainbow plot for the caliper and bridge geometry. The results show slightly lower Von Misses stress levels since the bridge was increased to its system limits. The resulting maximum Von Misses stresses of this updated analysis at the bridge envelope were in the order of 165-198MPa.

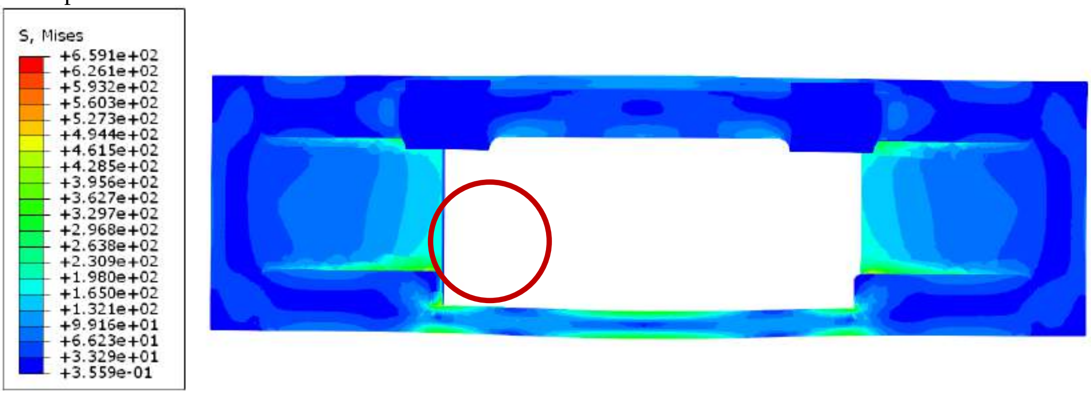


Figure 6: Slightly lower stresses present in the initial state of the optimization model (underside shown)

Next, a topology optimization was created using TOSCA, which is embedded within ABAQUS CAE. The objective function was to minimize the bridge volume, while constraining that the maximum Von Misses stresses could not exceed 255MPa. Additionally, a geometric restriction was added to ensure symmetry between the two caliper bridges.

After roughly 20 iterations, the solution converged (i.e. the differential from one run to the next fell below a configurable threshold), with a final bridge volume of roughly 38% of the original and a final maximum stress of ~250MPA within the bridges. Figure 7 shows the progress of the iterations to the optimized state.

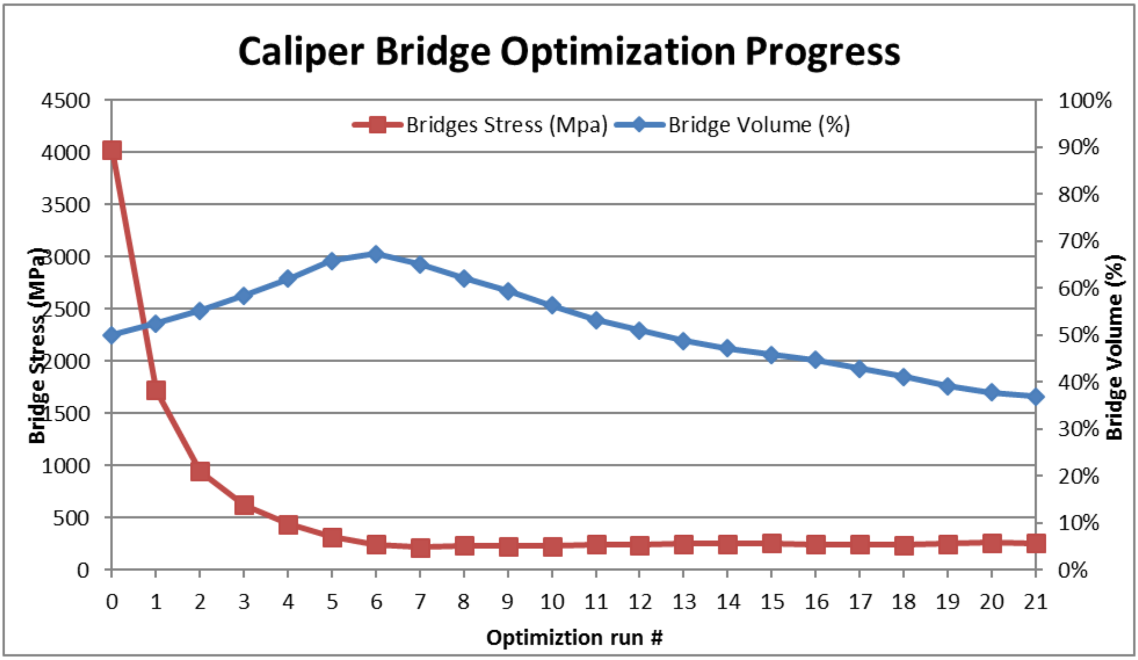
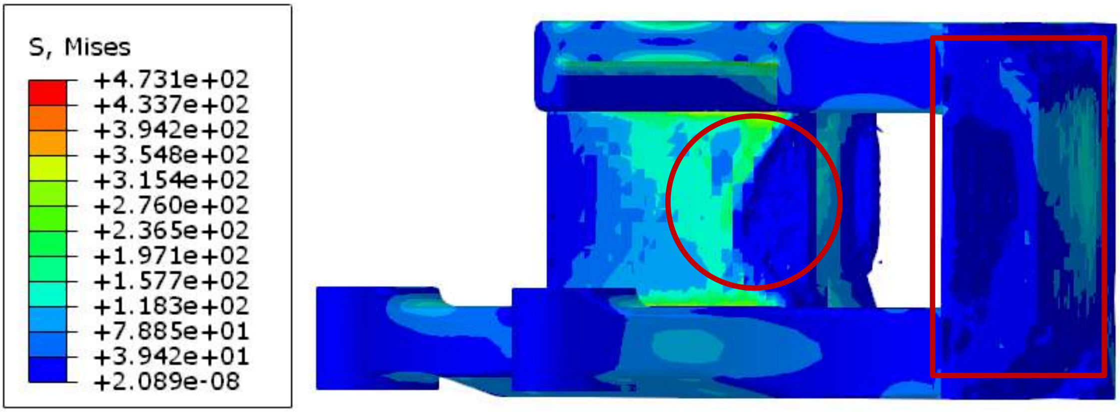


Figure 7: Progression of the bridge volume and stress during the optimization iterations

D. RESULTS

ABAQUS CAE was used to post-process the topology optimization results. Figure 8 shows the resulting topology-optimized bridge geometry and resulting Von Mises stresses of the bridge envelope. As seen, the stresses remained below the 255MPa constraint (197-235Mpa in the target area). The resulting bridge geometry volume was reduced by 62%, which represented a weight reduction for the brake caliper of more than 10%. TOSCA's reduction was achieved by hollowing out the bridge geometry, effectively resulting in a thin member across the bottom in tension and a slightly thicker member around the outside in compression (rectangle).



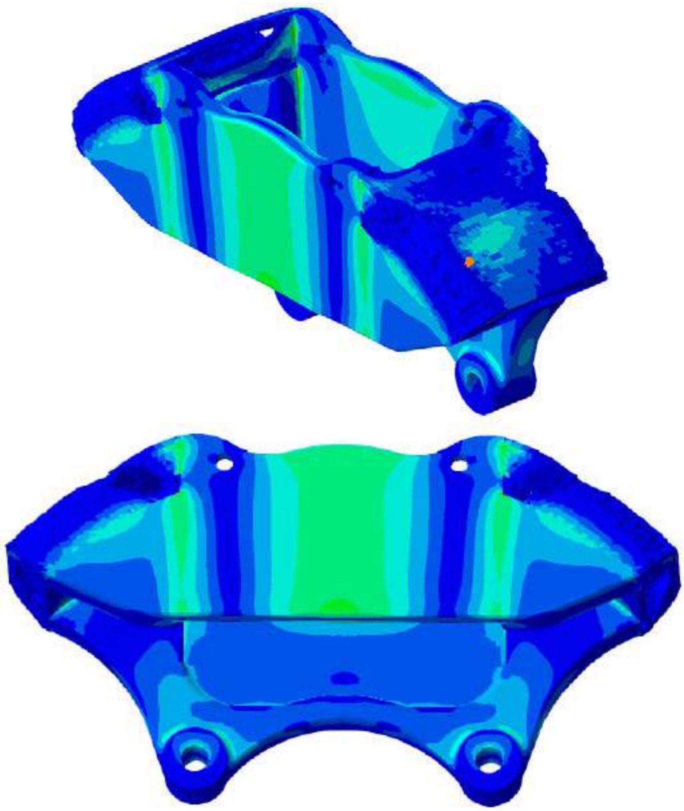


Figure 8: Various views of the optimized bridge geometry

Given the resulting amorphous topology optimized caliper bridge geometry, this part would be a good candidate for additive manufacturing techniques. However, given the conflicting requirements of high strength and low weight in this application, the manufacturing method in this case would most likely continue to be a casting operation. Additive manufacturing could potentially be leveraged for the toolmaking, but traditional methods may still prevail in that case. Now that we had a topology optimized geometry for the caliper, we needed to recreate it in CAD with traditional parametric features, such that it could be used in downstream processes (configuration modifications, manufacturing drawings, etc.). Figure 9 shows the resulting CAD geometry recreated from the topology optimized FEA model.

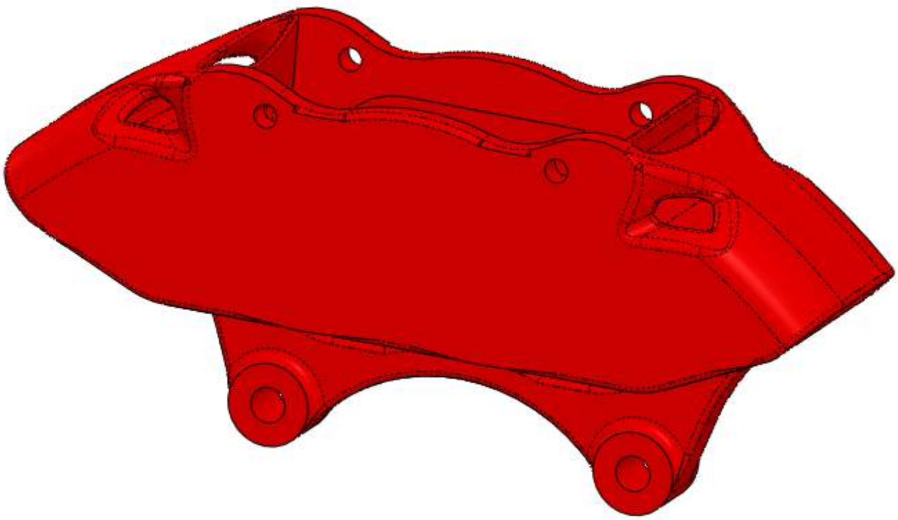


Figure 9: Parametric CAD approximation of non-parametrically optimized Tosca model

Once the CAD geometry was recreated, we needed to verify that these minor modifications did not somehow affected the resulting Von Misses stresses and the caliper bridge still meet the maximum stress constraint. Therefore, one last analysis was performed in ABAQUS using the same process of importing the part and building the equivalent analysis model and getting results as shown previously. Figure 10 shows the Von Misses stress rainbow plot of the sanitized topology optimized caliper-bridge geometry. As seen, the final Von Misses stress levels are very similar to the previous topology optimized values.[1] The caliper bridge Von Misses stresses are in the order of 183-229MPa.

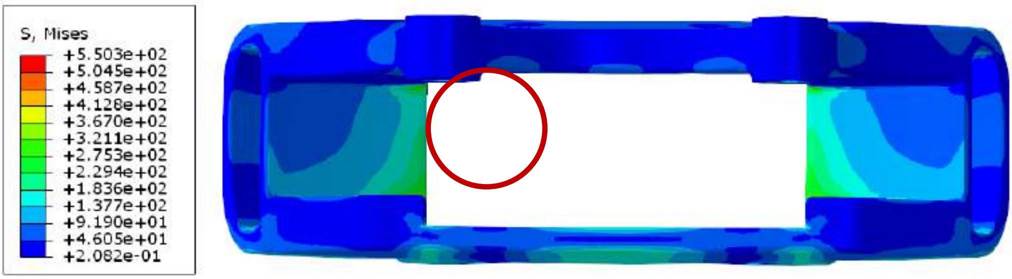


Figure 10: Final analysis results of parametric model in close agreement with optimized TOSCA model

[1] Note that the values although similar are not the same, which is why this final step should be performed.

E. CONCLUSION

This example illustrates the topology optimization of the bridge region of a brake caliper. The results show that significant volume (in this case 62%) and weight reductions (in this case 10%) can be achieved through the automated non-parametric TOSCA algorithms available within ABAQUS CAE. This example shows that the procedure used is also simple - where the engineer uses the unified ABAQUS CAE interface to prepare and post-process the analysis model, as well as prepare the model to be topology optimized using TOSCA embedded within ABAQUS CAE.

2. HIP IMPLANT NECK OPTIMIZATION

A. INTRODUCTION

Prosthetic devices are often made from metal alloys for strength and durability, and as a result can be heavier than the original body parts they are replacing. As a simple optimization scope, the neck of a prosthetic hip implant is targeted to remove weight from it while maintaining sufficient strength during the most extreme of its static loading cases.

In this case, a prosthetic hip was optimized specifically in its neck region, which joins the stem (inserted into the femur) and the femoral head (which mounts into an acetabular shell that’s set within the pelvis). The optimization was set to reduce the overall volume of the neck while a) preserving the mounting regions for neighboring components, and b) not exceeding a Von Misses stress level of 225MPa, which was less than the yield stress of 275MPa for the neck material (annealed 302 stainless steel). This example discusses the process to build this optimization study and the results obtained.

B. INITIAL FINITE ELEMENT RESULTS

The topology of a hip implant was optimized, specifically in its neck region, which joins the stem and the femoral head. The process involved isolating the neck from the implant assembly (SolidWorks was again used), exporting to a neutral format (e.g. STEP), then importing the geometry and finishing the analysis model within ABAQUS CAE. Figure 11 below shows the hip implant assembly, and the neck component, highlighted in red.

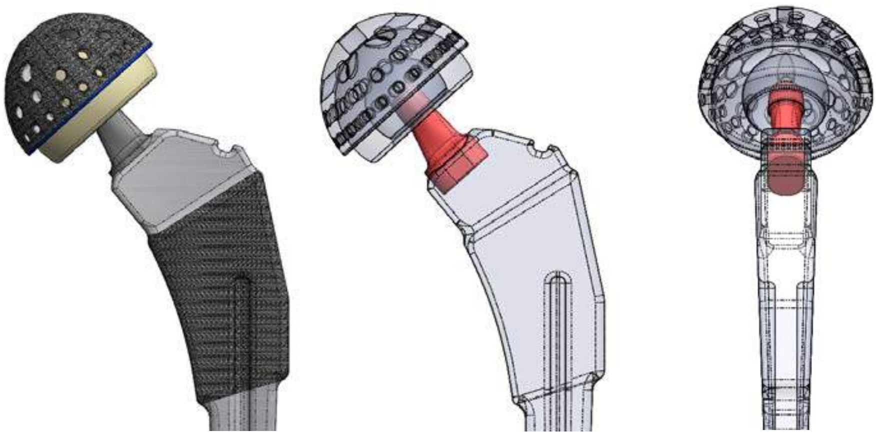


Figure 11: CAD model of hip implant, showing neck component targeted for optimization (in red)

To replicate the loading condition in Figure 12 below, two parts were used in the ABAQUS assembly. The femoral head was assigned material properties for polypropylene and the femoral neck was assigned material properties for annealed 302 stainless steel, and tie points were established between them, effectively bonding them together. The outer surfaces of the femoral head were set as a fixed boundary condition, and a concentrated force load of 1000N was applied laterally to a fixed region at the base of the femoral neck (where it is inserted into the stem).

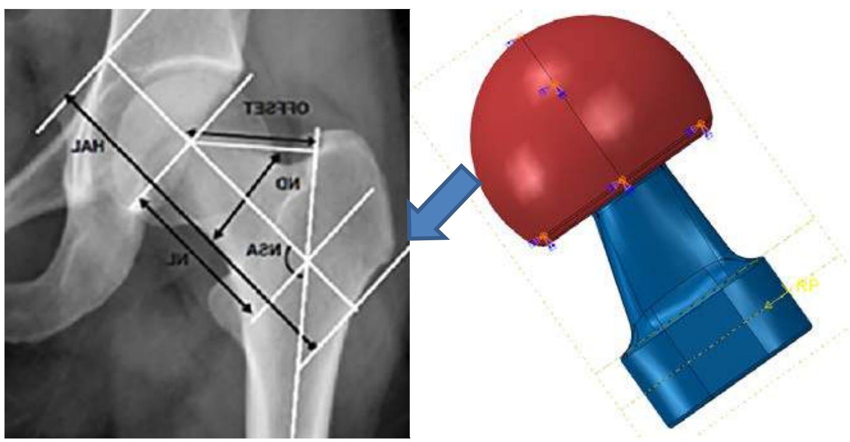


Figure 12: X-ray of hip joint showing lateral load condition (left), and analogous ABAQUS model used (right)

The initial results revealed excessive stresses (~360MPa) in the reduced region of the femoral neck, exceeding the yield strength of the material by roughly 30%. As seen below in Figure 13, this clearly indicates the original design's inability to withstand the lateral load case as prescribed, which can lead to fractures that have been seen in clinical conditions.

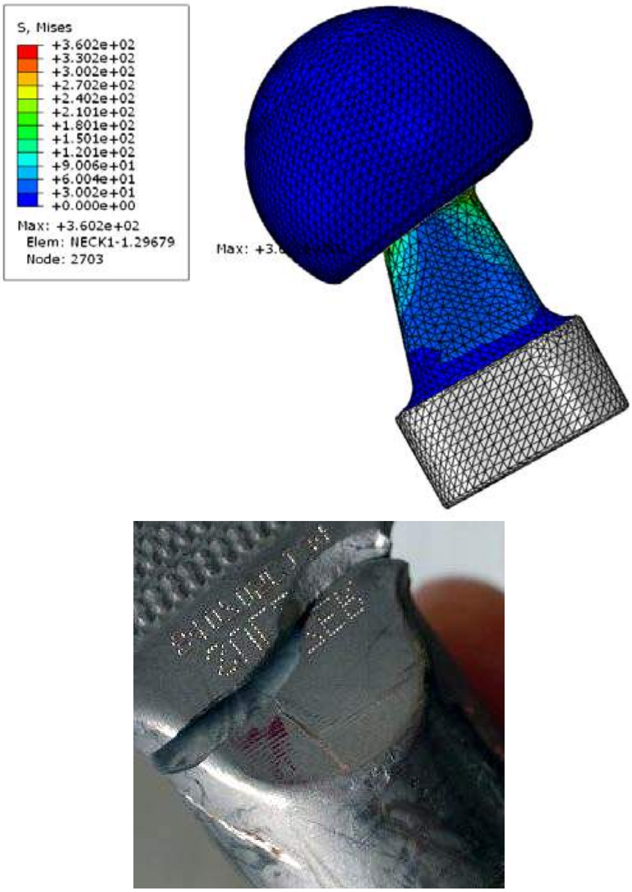


Figure 13: FEA results of initial design showing a peak of ~360MPa near the top of the neck (left), clinical failure of a femoral stem (right)

C. TOPOLOGY OPTIMIZATION

Using the same analysis conditions as above, a nonparametric topology optimization of the femoral neck was performed using TOSCA, again directly from within ABAQUS CAE. In this case, the design space was the section of the femoral neck between the femoral head and the stem.

However, like the caliper example, we first increased the volume of the neck slightly to its maximum extent while still maintaining clearance during joint articulation. This was done to a) eliminate the stress concentration regions of the original design noted above, and b) to provide as much material volume as possible for the optimization algorithms to utilize. Figure 14 below shows the imported femoral neck CAD model after this envelope change, and then after partitioning in ABAQUS to isolate the design space within the middle of the part.

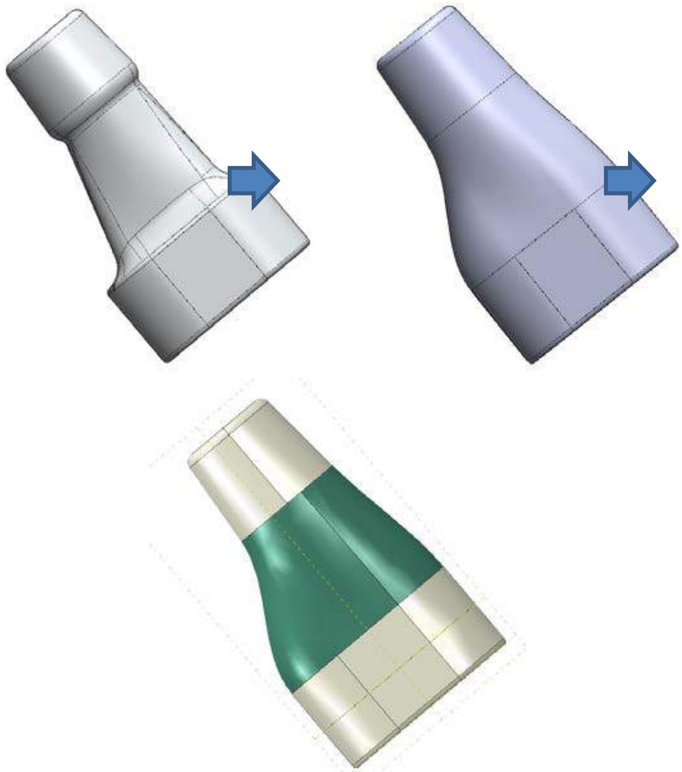


Figure 14: Original femoral neck CAD model (left), increased size CAD model (middle), and partitioned analysis model showing design space in green (right)

The same analysis conditions were set for the updated geometry, using the outer surfaces of the femoral head as a fixed boundary condition and applying a 1000N concentrated lateral load on the fixed region of the femoral neck base, as shown below in Figure 15.

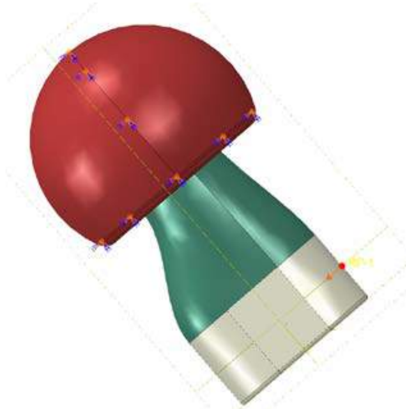


Figure 15: Loading conditions for optimization case, again with a rigid section at the base

Prior to running the iterative optimization, another single-step analysis was performed to double check the conditions and see if there was any improvement over the original design above. Results in Figure 16 revealed a substantial improvement over the original femoral neck model above, with stresses of ~190 MPa within the neck, nearly 50% lower than previously. With this good baseline condition, the optimization could now be run.

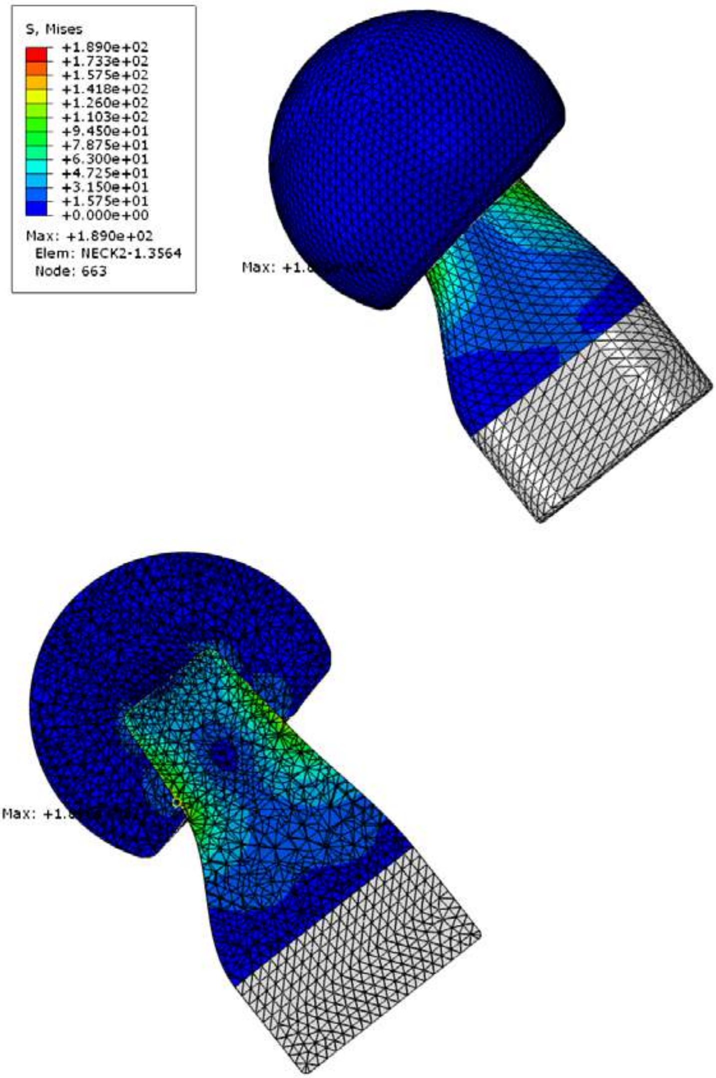


Figure 16: FEA results of updated, pre-optimized design, showing stress peaks of ~190MPa within the femoral neck

Next, the TOSCA topology optimization was created within ABAQUS CAE. The objective function was to minimize the neck design space volume (in green above), while constraining that the maximum Von Misses stresses could not exceed 150MPa (within that region). Additionally, a geometric restriction was added to ensure Z-plane symmetry of the neck (between the front and back in these views).

After roughly 20 iterations, the solution converged (i.e. the differentials from one run to the next fell below a configurable threshold), with a design space volume of roughly 52% of the original and a final maximum stress of ~150MPA within the design space, and ~226MPa within the entire neck itself. Figure 17 below shows the progress of the iterations to the optimized state.

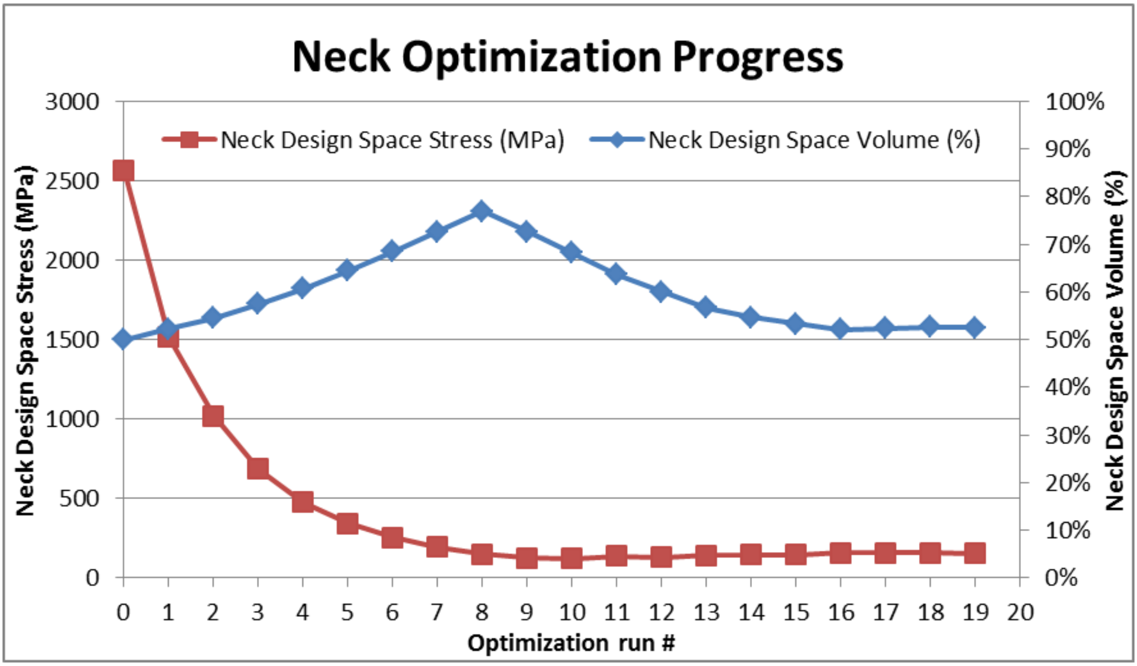


Figure 17: Progression of the neck design space volume and stress during the optimization iterations

D. RESULTS

ABAQUS CAE was used to post-process the topology optimization results. Figure 18 shows the resulting topology optimized femoral neck geometry and resulting Von Misses stresses of the entire neck component. As seen, the stresses of ~226MPa remained well below the 275MPa yield strength. This 38% stress reduction (from the original design) was achieved while also reducing the design space volume by ~50% and reducing the overall neck weight by 10%. This strategic volume reduction was achieved by hollowing out parts of the neck geometry, effectively eliminating material in areas of low stress, and keeping it in areas of high stress.

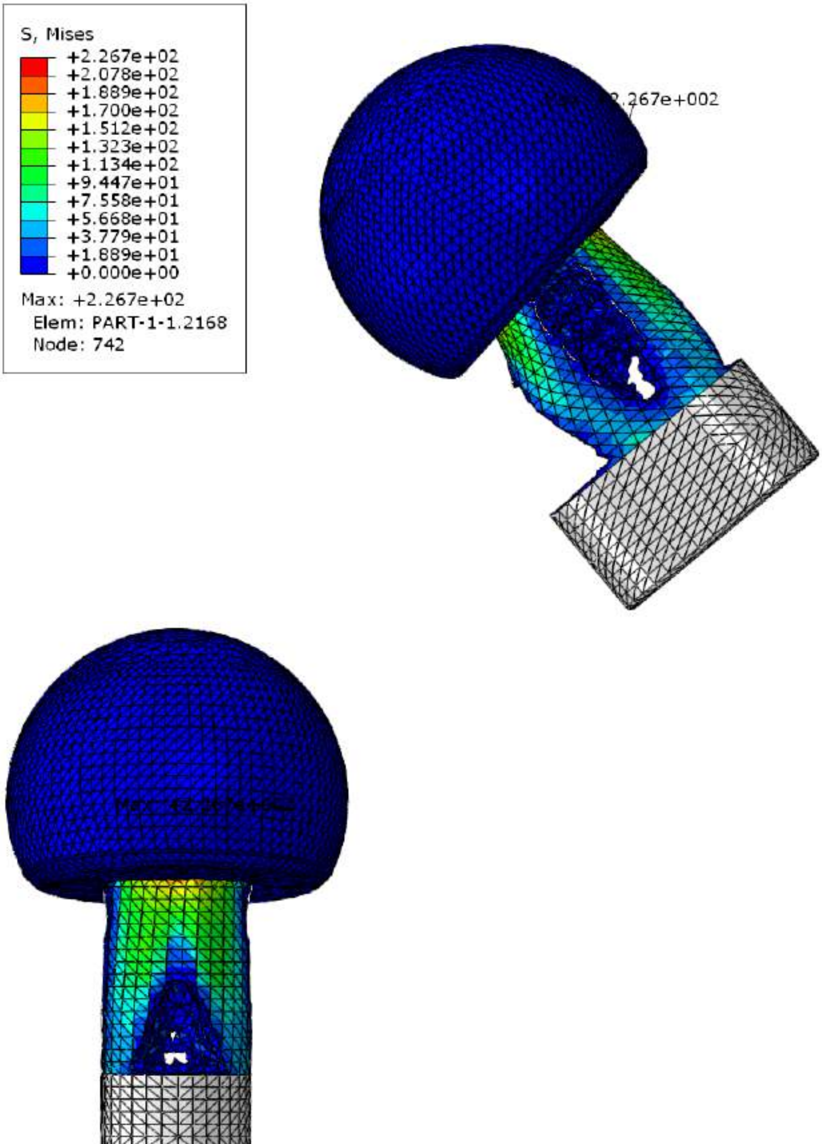


Figure 18: Results of TOSCA topology optimization, showing stress levels of ~226MPa within the femoral neck

As in the first example, despite being a strong candidate for additive manufacturing, we parameterized the model again in CAD for manufacturability of various hip implant configurations. Once the geometry was recreated, we again wanted to verify the targeted performance metrics in CAE. So, a final analysis was performed in ABAQUS using the same process of importing the part, building the equivalent analysis model, and performing the analysis. Figure 19 below shows the steps involved:

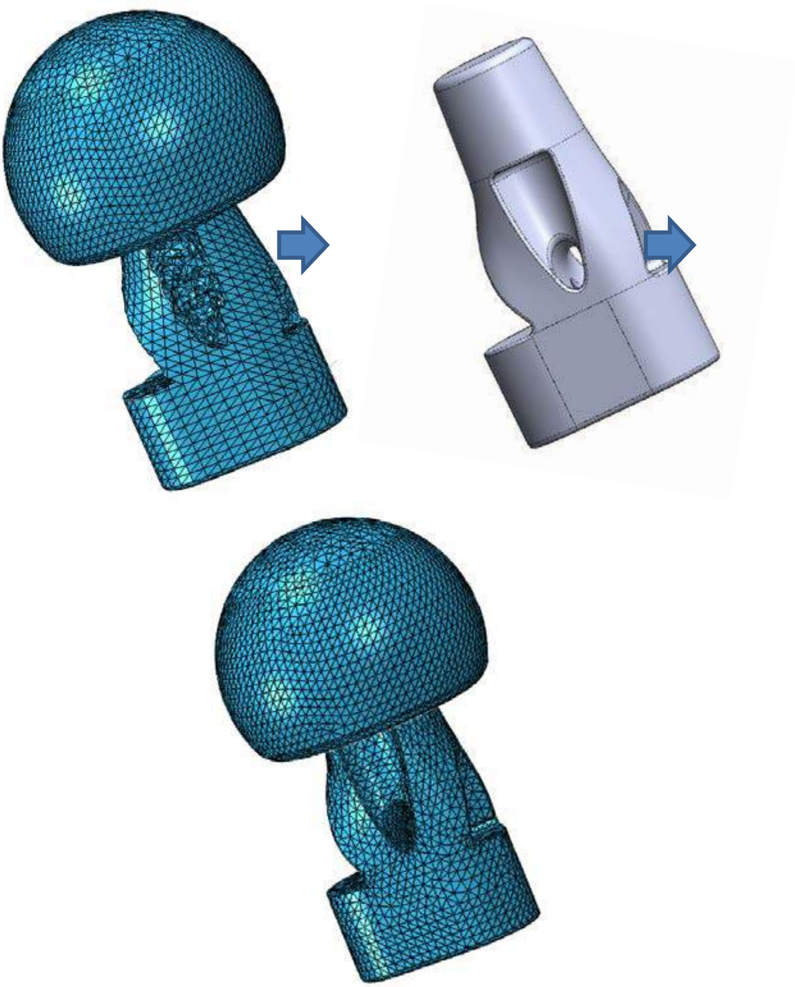


Figure 19: TOSCA-optimized model (left), CAD-generated equivalent (middle), and imported ABAQUS model (right)

Figure 20 below shows the Von Misses stress rainbow plot of the sanitized topology optimized femoral neck geometry. As seen, the final Von Misses stress levels are similar to the previous topology optimized values, in fact a little less at 208MPa, within 8% of the TOSCA-optimized results above.

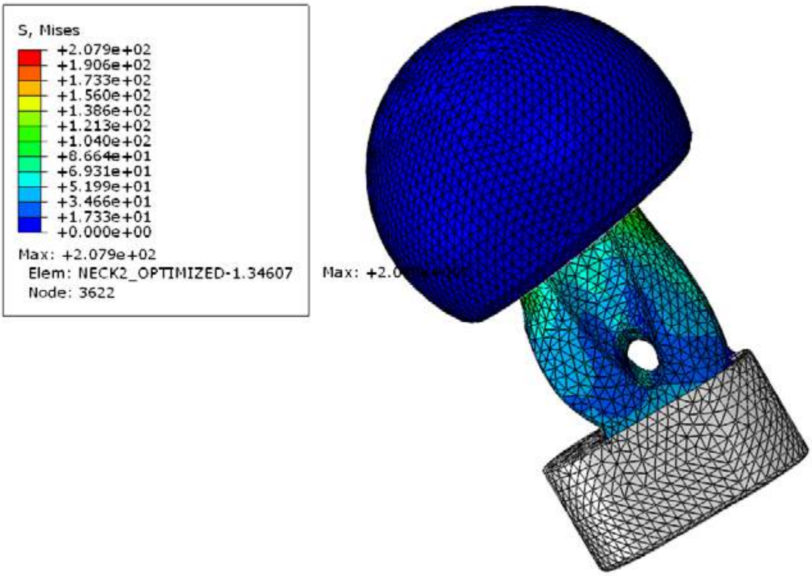


Figure 20: Final analysis results of parametric model in close agreement with optimized TOSCA model

E. CONCLUSION

This second case shown illustrates the topology optimization of the femoral neck region of a prosthetic hip implant. The results show that significant volume (in this case 50%) and weight reductions (in this case 10%) can be achieved through the automated non-parametric TOSCA algorithms available within ABAQUS CAE. Again, this case shows that the procedure is also very simple - where the engineer uses the unified ABAQUS CAE interface to prepare and post-process the analysis model, as well as prepare the model to be topology optimized using TOSCA embedded within ABAQUS CAE.