Topology Optimization of a Lacrosse Head

Rob Stupplebeen

President of Optimal Device

Abstract

Like many industries the balance between cost, stiffness, strength and weight is critical in sports equipment. This study goes through the methodology to perform topology optimization with Catia (CAD), Abaqus (FEA), Tosca (Topology Optimization) and Simpoe (Injection Molding). Topology optimization evolves the geometry to remove unneeded material effectively minimizing weight and maximizing performance. This is carried out by automatically scaling individual element's density and stiffness based on the stress state of the previous simulation. This is an iterative process where material flows to regions to satisfy constraints and minimize the objective function.

Keywords

Composites, Contact & Impact Mechanics, Design Optimization, Manufacturing, Material Modeling, Optimization, Process Automation

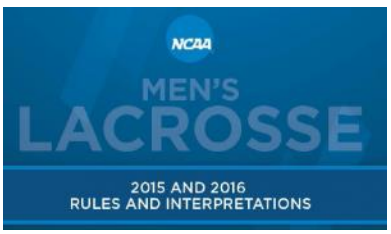
Introduction

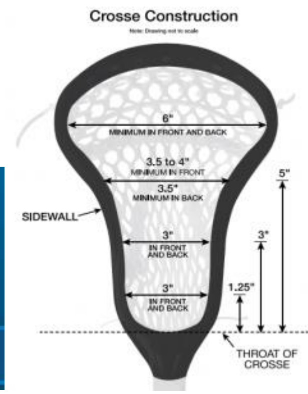
It doesn't matter what industry you are in common design objective are better, faster and cheaper. Lacrosse was invented by the Native Americans and involved a stick with leather woven net at the end to catch, hold and throw a ball. The equipment has evolved to a high performance alloy shaft usually a blend of aluminum, titanium and other metals. The head, which is the focus of this paper, is typically injection molded nylon66. The sport of lacrosse has claimed the tagline of 'the fastest sport on two feet'. It's a sport with a goal similar to soccer or hockey with similar rules. The lacrosse head is used to catch, throw, carry and pick up the ball. In addition to this the stick is used in hitting the opposing player if they have the ball. The head needs to be: stiff, durable, light and cost competitive.

This paper is going to go through the workflow to create a topologically optimized design. The process begins by creating geometry appropriate for topology optimization in CATIA. Then an Abaqus model is created with various loading scenarios and manufacturing techniques specifically injection molding and additive manufacturing. Apply appropriate controls for the topology optimization with Tosca. Then finally simulate the manufacturing process. For injection molding with Simpoe-Mold and for additive manufacturing an in development Abaqus plugin has been evaluated.

Geometry

To create the geometry the NCAA Men's Lacrosse rule book was used as a reference. A commercially available lacrosse stick was also used along with my trusty calipers.



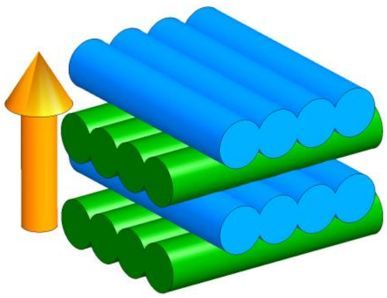


With this information a CAD model was created in CATIA using the Generative Shape Design (GSD) workbench to create the freeform parametric surface based features. Due to the audience the details of the geometry creation will not be covered in this paper but has been detailed in my blog at http://optimaldevice.com/blog/lacrosse-head-topology-optimization/ which includes the CAD and FEA models. This geometry reflects the largest feasible design envelope. This is important so that the topology optimization has the full design space to work with to erode away material where it is not needed.

Materials

Traditionally lacrosse heads are injection molded out of a toughened nylon66. www.matweb.com was referenced for material properties of nylon 66 inputting an elastic plastic material model into Abaqus of: E=2220MPa, nu=0.35 and yield stress of 65.8MPa.

Additive manufacturing and specifically Fused Deposition Modeling (FDM) have a structure similar to composite materials which have a long history being modeled with Finite Element Analysis.



A set of anisotropic material properties for 3D printed ABS were found in Monish Shivappa Mamadapur's paper CONSTITUTIVE MODELING OF FUSED DEPOSITION MODELING ACRYLONITRILE BUTADIENE STYRENE (ABS). This work assumes that you can homogenize the material properties of several layers and that the build direction will have a different set of properties. Again, this is standard practice for fibrous composite laminates.

These properties are listed as:

E2 = E1 = 1636 MPa, E3 = 1197 MPa

nu21 = 0.39, nu31 = nu32 = 0.37

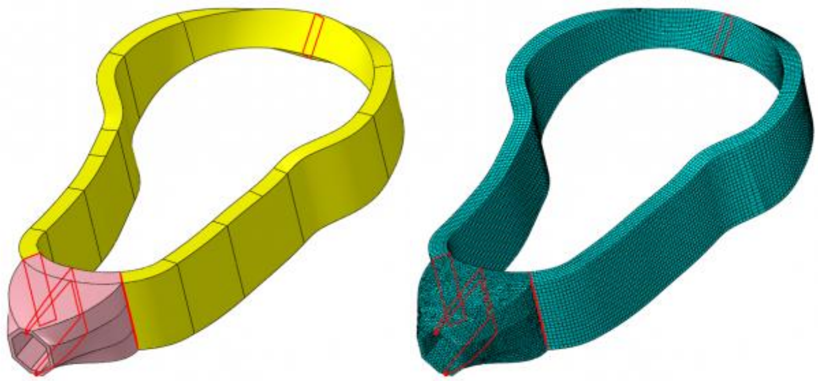
G13 = G23 = 645 MPa, G12 = 676 MPa

Abaqus however requires the constants for the nu's to be in the opposite convention. From efunda a conversion is provided nu12=nu21/E2*E1 which converts to:

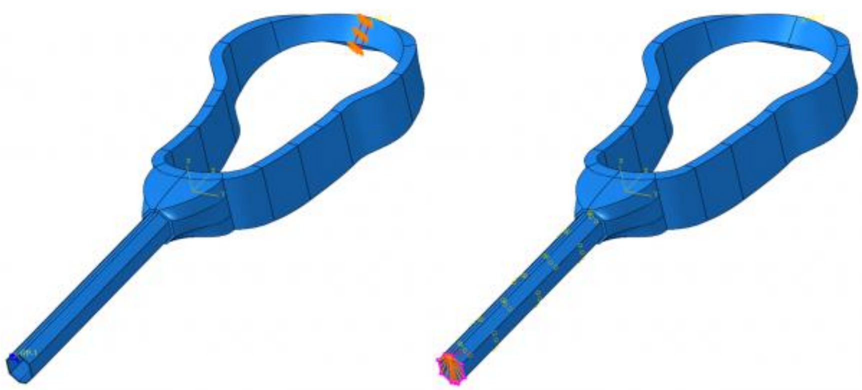
nu12= 0.39, nu13=nu32=0.51

Analysis FEA

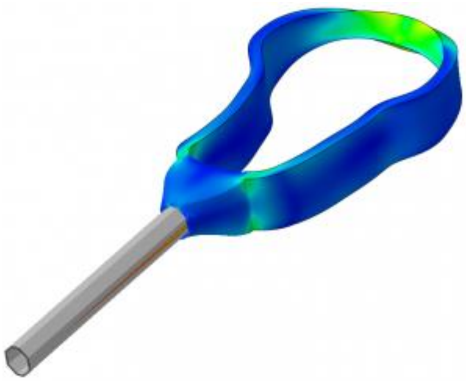
Upon importation into Abaqus, by way of a neutral STEP file, the lacrosse head geometry was partitioned with the red cut planes. This allowed for a hexahedral mesh in the yellow regions which are much more computationally efficient than tetrahedral meshes. The pink region was a more complex shape where a computational hit was accepted to reduce the user time vs. creating a hexahedral mesh.

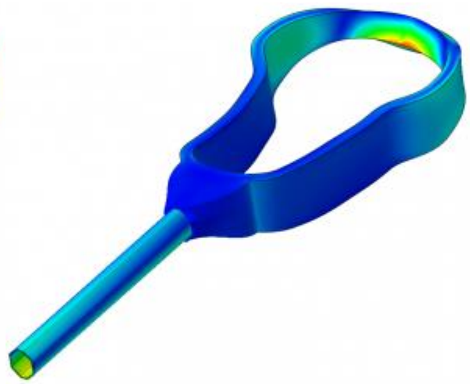


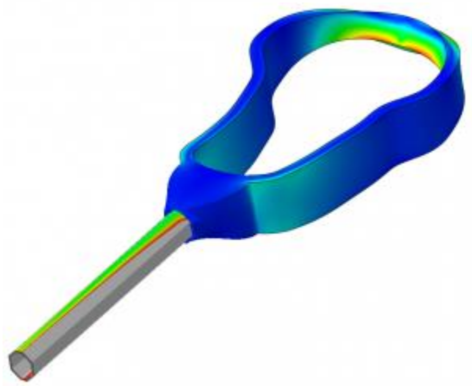
The cut plane at the top of the head was fixed to provide a load sink. Coupling constraints were added to the bottom of the shaft to a reference point. The shaft was tied to the head for load transfer. Rotations were fixed and 3 load cases were created for applying force in the orthogonal directions. An initial simulation was used to scale the loads to achieve a similar level of maximum stress in the head. This was done to drive stiffness in each direction similarly. These loads could be scaled to represent expected design loading and customized for player position or skill level.



Baseline results of stresses plotted on the same scale. The loads were: top, side and front respectively.





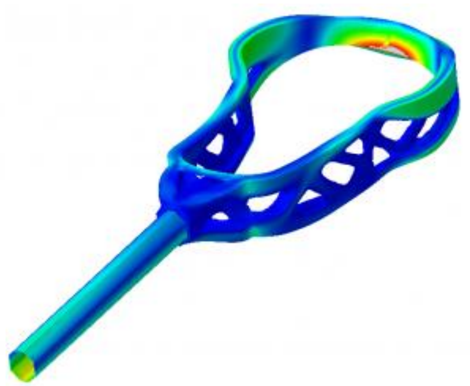


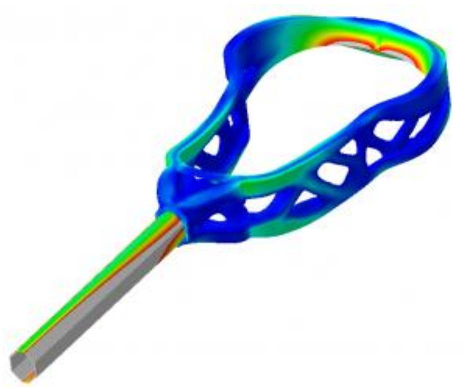
Topology Optimization

A topology optimization was created in Abaqus/CAE which uses Tosca in the background for the optimization. The optimization minimized strain energy and targeted a 50% mass reduction as a constraint. Geometric restrictions included: a size restriction for a minimum cross-section, symmetry and freezing the faces in red. Optimized designs must be manufacturable. Tosca provides tools to assure various manufacturing technologies are viable. In the case of lacrosse heads they are traditionally injection molded out of nylon. Injection molding has requirements such as draft and pull direction which must be satisfied to produce a cost competitive product.

Here is the optimized geometry after 36 design cycles with a clock time of roughly 5 hours on a modest CAD laptop.

Optimized results of stresses plotted on the same scale as previously. The loads were: top, side and front respectively.



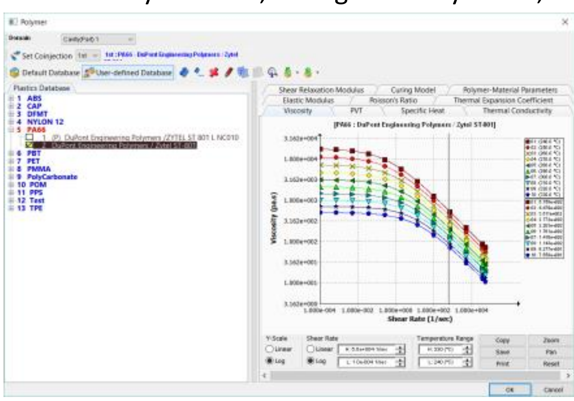


Exporting and Importing Tosca Results

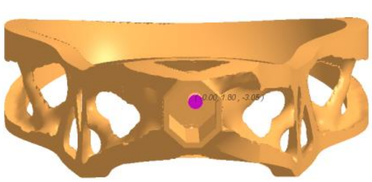
Optimized geometry can be extracted from the results from within Abaqus/CAE in either STL or INP format. The optimized geometry was imported into Catia by creating an STL file and using Digitized Shape Editor for point cloud manipulation. Quick Surface Reconstruction or Generative Shape Design can be used to create NURBS geometry for further design work. This is again outside the scope for this audience.

Injection Molding Simulation

Lacrosse heads are currently injection molded therefore Simpoe-Mold has been used to simulate the flow pattern of this new design. Inside Simpoe-Mold the STL was imported and meshed with shell elements. Zytel ST-80, a toughened Nylon 66, was chosen for the polymer.

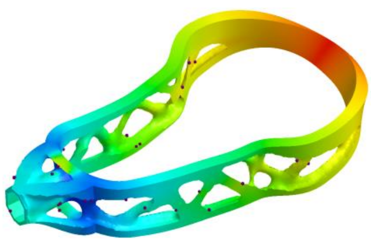


The gate location was chosen to be where the shaft assembles into the head. This is meant to hide the blemish caused by the gate on the finished product. On several commercial heads there are 2 gates located on the inside of the sidewalls instead. In Simpoe-Mold the gate location is chosen by screen selecting a point in the geometry as shown below.

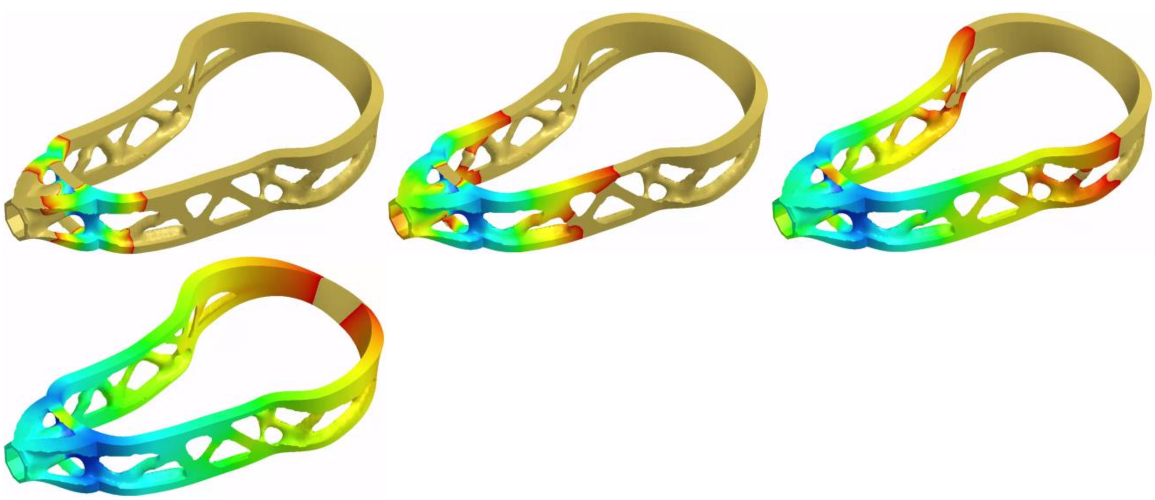


A lightweight preview of the simulation can be seen by selecting ‘Predict Flow Pattern’ which provides a rough estimate for the flow. The cool colors show where the flow initially enters and the warm colors show the last to fill material.

The full simulation shown below agrees fairly well with the initial prediction and can be see below. Undesirable air traps shown at the dot locations.



The flow front is shown at various stages of the fill process similar to molding a ‘short shot’ where the filling phase is stopped midway through the injection molding as a diagnostic tool.



It is clear that the gate location or the geometry will need to be reevaluated. There are several knit lines and air traps which will cause undesirable performance and aesthetics.

Topology Optimization for 3D Printing

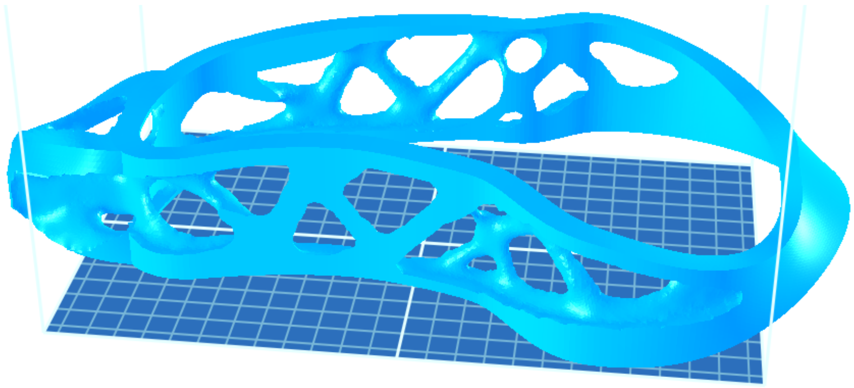
Topology optimization organic designs are ideally suited to the minimal manufacturing constraints that 3D printing offers. 3D printed parts by virtue of their layer by layer additive manufacturing approach have complex material properties. These properties are similar to wood where there is a stiff direction (with the grain) and a weak direction (across the grain). To gain the highest performance in 3D printed parts these material properties must be considered in the design process and included in the finite element model and optimization process. Additional optimizations were performed with the FDM material properties which were detailed previously. The topologically optimized geometry is shown below with the injection molded design first and then with FDM build directions in the X (front), Y (side) and Z (top) respectively.

As you can see all of these designs form a classical truss structure with diagonal supports connecting the front and back of the head. There are subtle differences between these upon further inspection.

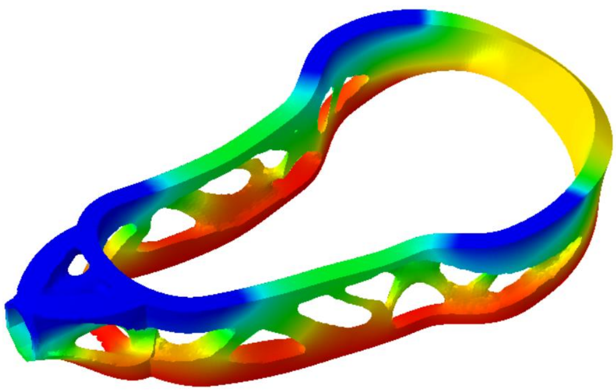
Additive Manufacturing Simulation

Additive manufacturing in general applies layers of hot material which cools and hardens while the next layer is added. This process locks in thermal stresses. These stresses can cause part warpage, reduced ultimate load or reduced fatigue life. A workflow in development by Simulia addresses this process. This workflow uses Abaqus for a sequentially coupled thermal and structural analyses automated with scripting in Python.

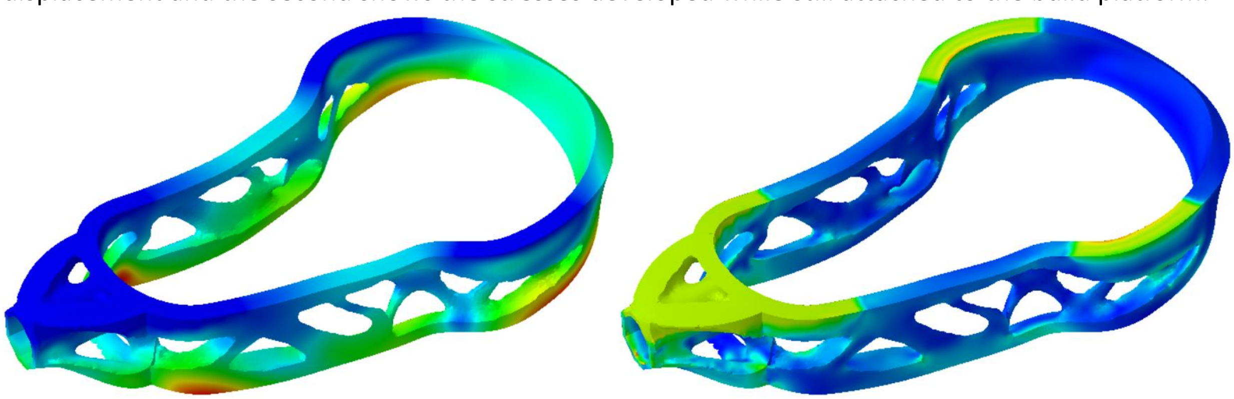
G-code has been created based of the STL of the topologically optimized geometry using a free tool path generator, ReplicatorG. This G-code is used by the Abaqus scripts to assign the status of the elements with respect to time for the thermal and structural simulations.



A general heat transfer analysis is created in Abaqus/CAE. The plug-in reads in the G-code created by ReplicatorG. This is used to enable elements at the initial hot temperature. The heat is dissipated through the front face which is in contact with the build platform.



A general static analysis is created in Abaqus/CAE. The front face is assumed to be rigidly attached to the build platform. The predefined temperature fields are imported from the previous general heat transfer analysis. The plug-in again reads the G-code which when coupled with the heat transfer results simulates the thermos-mechanical stresses developed during manufacture. The first image shows the displacement and the second shows the stresses developed while still attached to the build platform.



Conclusion

This study has gone through the workflow to create a topologically optimized lacrosse head for both injection molding and for additive manufacturing. All designs are subjected to constraints and goals. Geometric constraints were addressed with Catia and Tosca. Performance metrics were simulated with Abaqus. Manufacturability has been simulated with Simpoe and the new additive manufacturing workflow within Abaqus.