Structural Optimization of a Cleat for a Springfree® Trampoline

Guido M. Quesada

CAE Analyst, Matrix Applied Computing, Auckland, New Zealand

Hamish McIntyre

Engineering Manager, Springfree NZ Ltd., Christchurch, New Zealand

Abstract: This paper reviews the structural optimization of a cleat from a Springfree® Trampoline. In this innovative trampoline, instead of using metal springs the mat is suspended on flexible rods attached to cleats made of fibre glass reinforced polypropylene. A physical testing device was built which applies the load from a rod at the centre of the cleat and supports it at the sides. Nonlinear simulations were used to replicate this experiment and evaluate the effect of changes in the cleat geometry before prototyping. The density and orientation of the glass fibre has an effect on the anisotropic material properties and orientations. Also residual stresses have an effect on structural performance. Therefore an injection simulation was performed using MoldFlow®. The results were imported into Abaqus/CAE to add the cleat to an assembly including two rigid surfaces which represent the testing device and the model was completed by editing the input file manually. This procedure was effective but too slow for an iterative optimization process. Therefore average isotropic material properties were estimated and modal eigenfrequency was chosen as the design response to maximize. A spring-mass analogy was used to relate stiffness to natural frequency and mass, particularly for a mode that resembles deformation under the physical test. Two other deformation modes were evaluated as well which provide an idea of the overall robustness of the cleat. Once a series of iterations lead to a satisfactory improvement in stiffness and mass, the nonlinear simulation procedure was repeated to verify the results.

Keywords: Design Optimization, Modal Analysis, Abaqus Topology Optimization Module, Abaqus Interface for Moldflow.

1. Introduction

Springfree NZ Ltd design and manufacture a unique type of trampoline that does not require traditional metal springs. This greatly enhances the safety of the trampolines for home use by moving the frame away from the jumping surface and creating a soft edge to the mat. A typical unit is shown in figure 1. One element of the design is a set of glass-fiber reinforced polypropylene cleats. These cleats attach the trampoline mat to a series of rods which provide the spring effect. There is an interest in determining the stresses in the cleats during operation and the effect of design changes on their load bearing capacity.

Figure 1. Springfree Trampoline.

2. Problem specification

The specification of the problem evolved as the first results were found. Initially, a physical testing device was built to apply the load from a rod (similar to the ones used in the trampolines) at a socket near the center of the cleat. The cleat was placed upside down on a supporting base, as shown in Figure 2.

An early cleat design was analyzed by performing an FEA simulation of this experimental test. This early design was used as a baseline for the evaluation of design changes. A second version featured pockets in regions which were thought to be structurally neutral, thereby reducing the mass from 46.5g to 43.1g. The FEA model showed that this change had actually led to a reduction in the cleat strength. This established the framework for an optimization process. The objective was to maximize cleat load bearing capacity by adding material to the second version of the cleat (43.1g) and keeping its mass below 46.5g.

3. Methodology

A general design iteration step involves deciding which design changes to evaluate, solid modeling of the modified part, injection molding simulation of the part, and FEA simulation of the physical test. One difficulty was that Abaqus Interface for Moldflow did not provide a straightforward approach for the analysis of an assembly involving parts that were not in the injection molding simulation. Additionally the physical test simulation was relatively costly for multiple iterations, especially if Moldflow results were to be considered.

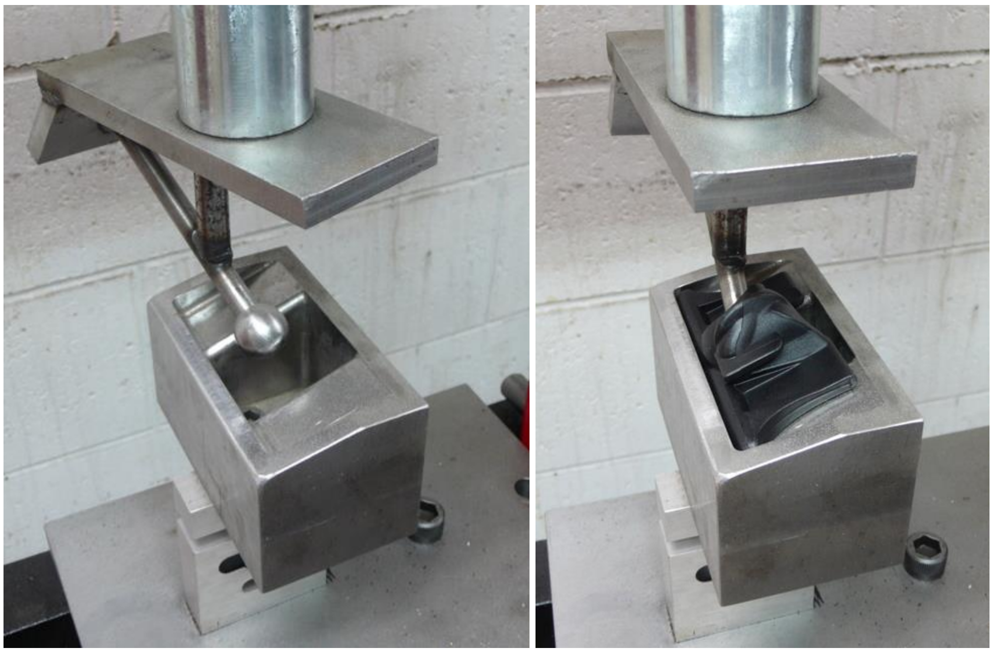

Figure 2. Physical test setting

In order to streamline the design process, it was agreed that tentative design changes would exclude a rerun of the Moldflow simulation. This simulation was included only for selected proposals which showed significant improvements according to relatively quick analyses.

The optimization process was initially based on engineering judgment of the results, without using the Abaqus Topology Optimization Module (ATOM). Then ATOM was available and it was possible to compare the results and evaluate the convenience of the automated optimization process. Finally, since not all design preferences were specified in ATOM, a final design interpretation was created and analyzed.

3.1 General FEA methodology

Nonlinear finite element analysis using Abaqus was used to predict the stresses in the cleats by replicating the physical test. The main nonlinear model incorporated the following features:

- Mechanical properties and initial stresses for an orthotropic material model were imported from a Moldflow mold injection simulation.
- Second order tetrahedral elements (C3D10) were preferred for final analysis, since they are more accurate in bending, which is the predominant deformation mode in this case. Linear elements (C3D4) were used to obtain faster results during design iterations which are shown in this paper.
- Nonlinear contact was modeled between the rod and the cleat and between the cleat and the base.
- A quasi-static nonlinear approach was used, considering that the load is applied slowly.
- The base and rod including the ball at the tip which engages with the cleat socket were treated as rigid bodies.
- The base was fixed. The rod was fully constrained with a prescribed displacement in the loading (vertical) direction. The cleat was free to move, driven by its interaction with the rod and the base.

3.2 Nonlinear model using the Abaqus interface for Moldflow

Moldflow Plastics Insight (referred to as Moldflow in the Abaqus framework) is an injection molding simulation software package produced by Autodesk, Inc. Both the mechanical properties and residual stresses resulting from this simulation are relevant for the cleat. It is found that the distribution of material properties vary considerably from the surface to the core and the residual stress is also very significant.

The Abaqus Interface for Moldflow translates finite element information from a Moldflow analysis into an Abaqus input file. The translator uses a set of files that are created by the Moldflow analysis. A typical workflow is detailed in Appendix 7.1.

Since the simulation involved an assembly of three parts (the cleat, the base and the pole) as shown in figures 2 and 4, it was advantageous to assemble them in Abaqus CAE. The cleat was imported from the input file created by the Abaqus Interface for Moldflow, the base was imported as a discrete rigid surface from a parasolid file and the pole was created as an analytical rigid surface. Then the parts were instanced into the assembly, moved into position to avoid initial contact overclosure and subjected to their corresponding prescribed displacements and boundary conditions. The simulation involves one preliminary step in which initial stresses reach equilibrium and generate some geometric distortion. In the main step the ball at the end of the pole engages with a socket in the cleat and pushes it against the base.

Importing the cleat part into Abaqus/CAE, creating new parts and generating a new input file for analysis generates a series of naming conflicts. Also the initial stress data from the imported input file is not really necessary in CAE and tends to slow down its performance. Therefore a special procedure was implemented in order to address those issues. The steps are shown in detail in Appendix 7.2.

3.3 Matching the material properties from Moldflow with average properties

One drawback of the nonlinear simulation was that it involved a slow workflow. A first step towards simplifying the procedure was to estimate average material properties and to neglect the state of residual stress from the injection molding process. One approach which proved effective was to create a model of a cleat with average material properties and adjust its elastic modulus until its most relevant natural frequencies matched those of a model with material data from Moldflow. It was necessary to add an estimated material density, which is not provided in the Moldflow output.

3.4 Using a spring-mass analogy to create a simplified linear model

While exploring eigenmodes to compare models with complete and simplified material properties, it was found that the third eigenmode from a linear frequency analysis resembles the deformation mode observed in the physical test. This would enable a comparative analysis for optimization purposes at a fraction of the cost of the nonlinear analysis.

By considering a single degree of freedom spring-mass system, it is possible to relate a change in stiffness to a change in natural frequency. Although considering the mass distribution may address the bending mode more rigorously, it was found that this analogy was quite adequate for comparison purposes and involved simpler calculations. From the basic relationship for a simple spring-mass system, it can be noted that an increase in the frequency would mean that the stiffness has increased or that the mass has been reduced. Looking at changes in the stiffness per unit mass would be useful to assess how efficient the cleat becomes in terms of delivering greater stiffness or requiring less material. This was defined as an equivalent specific stiffness:

$$\eta_k = \frac{k}{m} = (2\pi f)^2$$

where f is the frequency, k is the spring stiffness and m is the mass. It should be noted that the cleat could be made very efficient by taking out a lot of mass, but this would also make it weaker. By considering that the eigenmode of interest resembles the bending of a simply supported beam, it is possible to relate the stiffness k of the beam to a load P applied at the middle:

$$P = 6\left(\frac{S}{E}\right)\left(\frac{l^2}{h}\right)k = 6\left(\frac{S\rho}{E}\right)(l^3 b)(2\pi f)^2$$

where S is the material strength, E is the elastic modulus, ρ is the density, l is a characteristic length, b is a characteristic width, and h is a characteristic height of the equivalent beam. What this suggests is that as long as the material properties and the overall part dimensions remain the same, the load bearing capacity is proportional to the stiffness, and this in turn relates it to the natural frequency. These assumptions are reasonable considering that the expected change in the mass of the cleat is limited to less than 10%, while the overall length, width and height of the cleat remain constant.

In summary, by using the natural frequency of the bending mode as a reference, it was possible to estimate an equivalent average elastic modulus for the cleat material. Furthermore, by using the spring-mass analogy, it was possible to use the frequency analysis as a quick method to compare the improvement in structural efficiency and load bearing capacity. This applies to both the manual and the ATOM optimization procedures.

3.5 Manual optimization of the cleat

Significant geometric changes to the part (whose geometry is relatively complex) would need to be performed by the designer in a solid modeler outside Abaqus CAE. This put an important limit on the number of iterations that could be evaluated within the available time and budget. The number of changes that could be tested was limited to about 20 selected combinations of variations involving three main features:

a) Increasing the wall thickness around the socket.
b) A variable number of ribs of similar size and orientation across the pocketed regions.
c) Angled vs. flat bottom of the pockets.
d) Variable pocket depth.

Some of these variations were generated in batches, so they could be processed more efficiently in the FEA.

3.6 Optimization using ATOM

Since the optimization process involves adding mass, rather than removing it, an optimization domain was created, extending around the region where reinforcing ribs were desirable (see figure 3). This domain was trimmed in order to make sure that the reinforcements would not grow larger than what was feasible from a manufacturability and functionality perspective (the cleat must fit inside pockets around the mat without creating any stress concentration). It was added within Abaqus CAE, so no interaction with the designer was required at this stage.

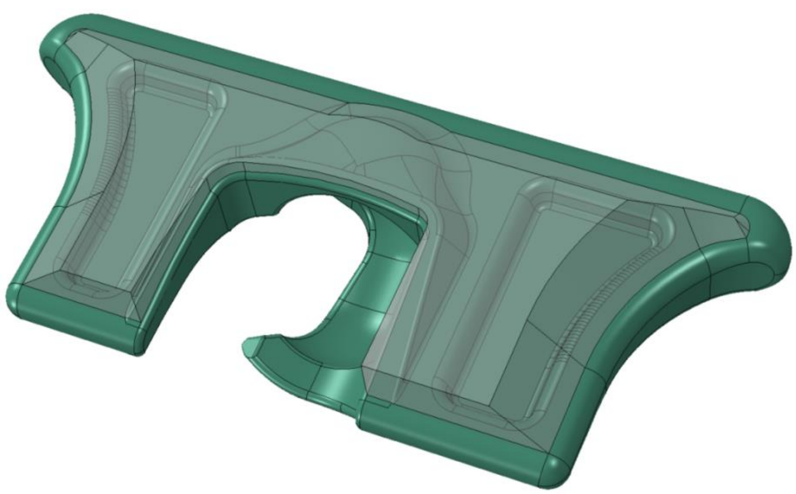

Figure 3. Domain for topology optimization shown as a transparent volume.

The design responses selected were the third natural frequency, which shows the bending mode, and the volume in the optimization domain. The objective function was defined to maximize this natural frequency. A more rigorous design response for this purpose would have been the frequency squared, or frequency squared divided by volume. However the natural frequency alone served a similar purpose, it was simpler and directly available in the ATOM interface.

The main constraint was to keep the volume of the optimization domain below a certain percentage, which was set so that the total mass of the cleat would remain below 46.5g. A geometric restriction was added using the "forging" technique with a 5° draft angle. This made sure that the added material would keep the cleat demoldable. Once the optimization job completed, the geometry was extracted and subjected to a frequency analysis, considering the ATOM density isosurface at which the part volume was trimmed.

4. Results

4.1 Nonlinear model using the Abaqus interface for Moldflow

Figures 5 and 6 show results of the test simulation including a deformable part with material properties and initial stress from Moldflow, with the pole and the base modeled as rigid surfaces. One distinctive aspect in these results is that the injection molding process tends to produce residual stresses which put the surfaces in compression (shown in dark grey in figure 4), while the core is in tension. This was a good reason to reduce the thickness of the part wherever possible. Another visible aspect is the irregularity of the stress distribution, which is inherited from the Moldflow results. Most importantly these results show the bending nature of the test and the distribution of maximum strains which should be reflected in the simplified approach.

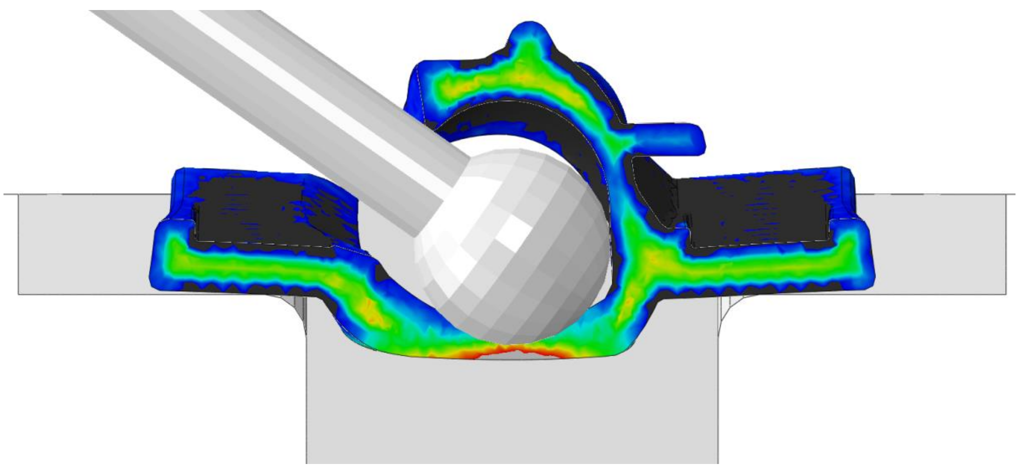

Figure 4. Maximum principal stress in a cross-section of a nonlinear load simulation using the Abaqus interface for Moldflow.

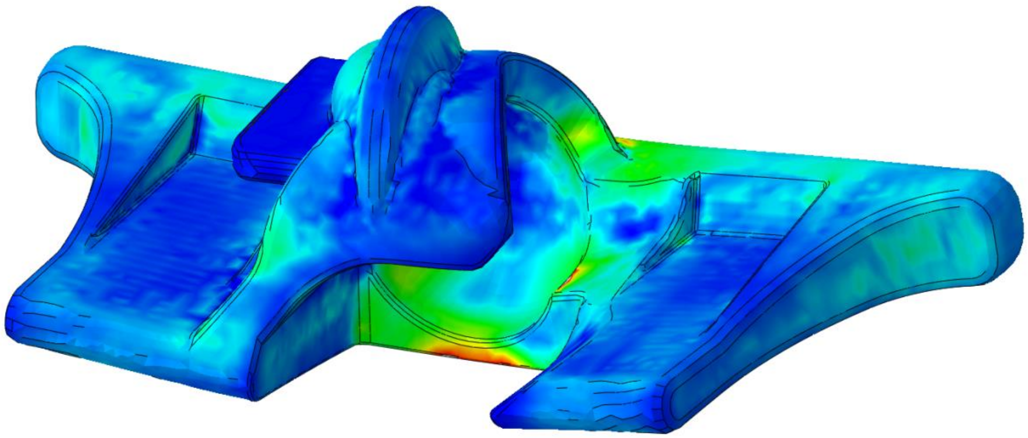

Figure 5. Maximum principal stress of a nonlinear load simulation using the Abaqus interface for Moldflow. Pole and base not shown.

4.2 Manual optimization of the cleat

One of the intermediate attempts of manual optimization is shown in figure 6. This involved 4 ribs on each side of the socket, extending along the pockets, which had a flat bottom. This version provided a 2% reduction in mass, with 14% increase in specific stiffness and 12% increase in absolute stiffness. The results were regarded as modest. It was also observed that there was poor of alignment of the ribs with a critical load path that runs along the top of the socket.

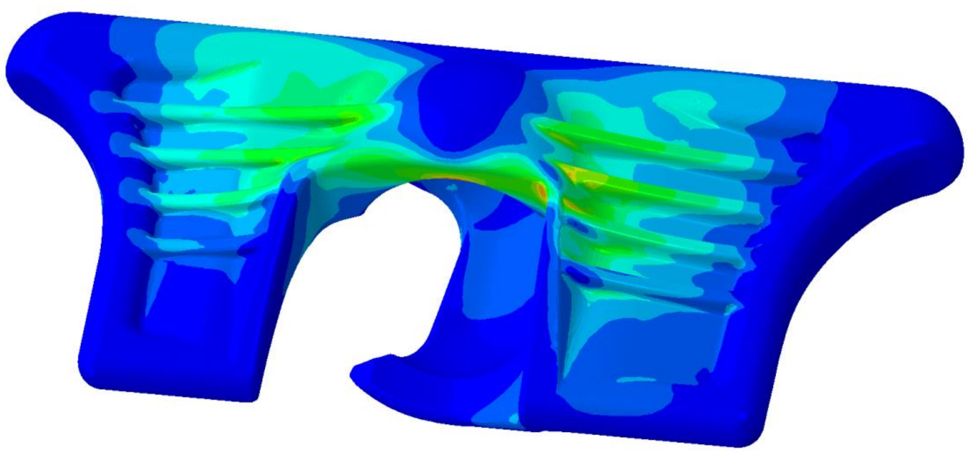

Figure 6. Strain distribution on the bending eigenmode in a design iteration using 4 thin ribs across the pockets on each side.

This led to another trial with a single thick rib across each pocket and angled pocket bottom. The result was a 20% increase in specific stiffness and a 21% increase in absolute stiffness, with the same mass as the baseline design. It was accepted as a successful trial and selected for final analysis using the nonlinear model with Moldflow simulation results. Incidentally the other two main deformation modes improved in stiffness as well.

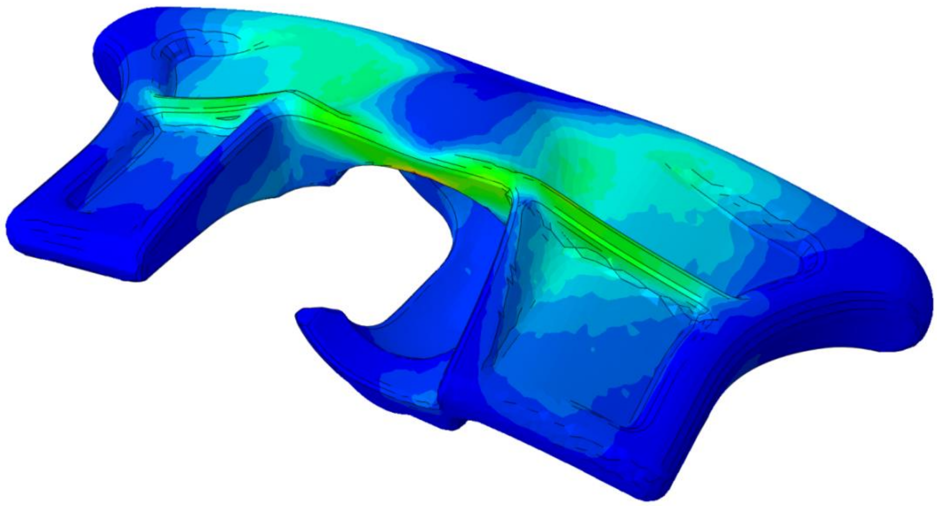

Figure 7. Strain distribution on the bending eigenmode in a design iteration using one thick rib across angled bottom pockets on each side.

4.3 Optimization using ATOM

The topology optimization finds the same two ribs which extend across the pockets from the sides of the socket which were found by hand, which is somewhat obvious. However the topology optimization also finds that these ribs should not be aligned with the longitudinal axis of the cleat. Instead they should diverge at an angle of about 5° as they extend from the socket. The added volume reaches the ceiling of the optimization domain above the top of the socket, where there is a weak spot. This was also identified during the manual optimization and led to the first changes in the design. Unfortunately it is not feasible to put more material there due to functional constraints.

The main feature that was missed in the manual optimization was a third rib extending from the socket to the main bar through a corner of one of the pockets. This feature is clearly needed to compensate the opening for the pole on the other side of the socket, but it did not become obvious until the ATOM results pointed it out. Most remarkably, the cleat volume (compared to the baseline design) was reduced by 4%, while specific stiffness increased 31% and absolute stiffness increased 25%. First and second mode stiffness improved as well, though not as much as in the manual approach.

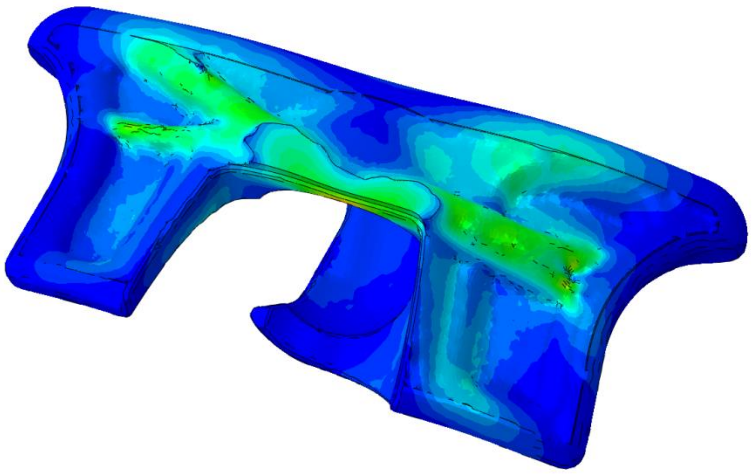

Figure 8. First result of topology optimization showing normalized strain in the bending natural mode.

Relatively thick features may not be as strong as predicted by the simplified analysis because they don't develop full strength at the core. Furthermore the rib on the right tends to become so wide that it splits, which is not a desirable feature. Taking this into consideration, an additional geometric restriction was introduced to limit member size to a maximum of 10mm. The result is shown in figure 9. While the ribs become thinner, they also become taller reaching the ceiling of the optimization domain. Despite the added constraint, this trial achieves a 41% increase in specific stiffness and 36% increase in absolute stiffness, still at a 4% reduction in mass. This was beyond any expectation during the manual optimization process.

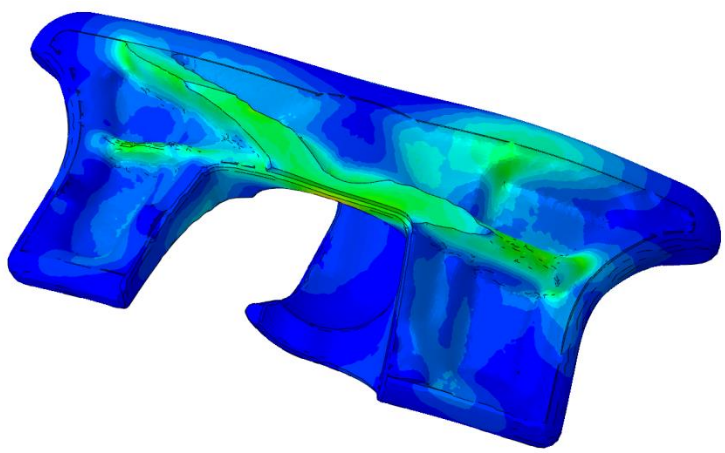

Figure 9. Second result of topology optimization showing normalized strain in the bending natural mode.

A third optimization trial was created with a more severe restriction on the member size but no relevant improvements were found.

In summary, by executing just three optimization jobs comprising a total of 110 iterations and requiring no intervention from the designer, ATOM was able to provide decisive pointers towards a final optimal design. This should be contrasted with about 20 manual iterations, each of them involving a separate analysis job, interpretation of results and new geometry created by the designer (though some changes were generated in batches to save time). While the manual process took place over several weeks (including communication and work delays), the ATOM process was completed in just a couple of days.

4.4 Designer's interpretation of ATOM results

The results from ATOM were used to build a solid model of a new design, which was then subjected to the simplified linear analysis. This design achieves a 29% increase in specific stiffness and 28% increase in absolute stiffness, at a 2% reduction in mass. Although the improvements are not as dramatic as in the ATOM results, they are still considerably better than in any previous design.

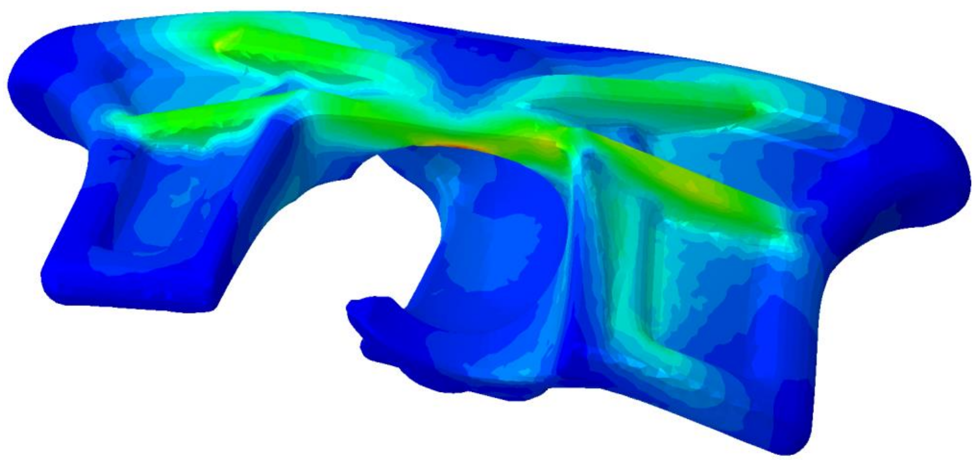

Figure 10. Normalized strain in the bending natural mode for designer's interpretation of ATOM result.

A fourth rib was added in order to make the part more symmetric. It makes it less efficient in bending, but in exchange it makes it considerably stronger in other deformation modes.

4.5 Final evaluation of improved designs

Unfortunately the results of final verification using the nonlinear model for the latest design based on ATOM are not available. However, given the results from the verification of manually optimized cleats it can be expected to provide even better results.

Figure 11 shows a Force vs. Displacement plot from the full nonlinear analysis, comparing an early design to selected finalist designs. The finalist designs are superior in terms of stiffness and strength. Notice the initial stiffness of these finalists is indistinguishable. Furthermore the one that finally appears to be stiffer turns out to exceed allowable stresses earlier than the more flexible alternative. In this regard it was found, for example, that the angled bottom pockets would make stiffer cleats, but this would also lead to greater stresses at weak spots away from the pockets.

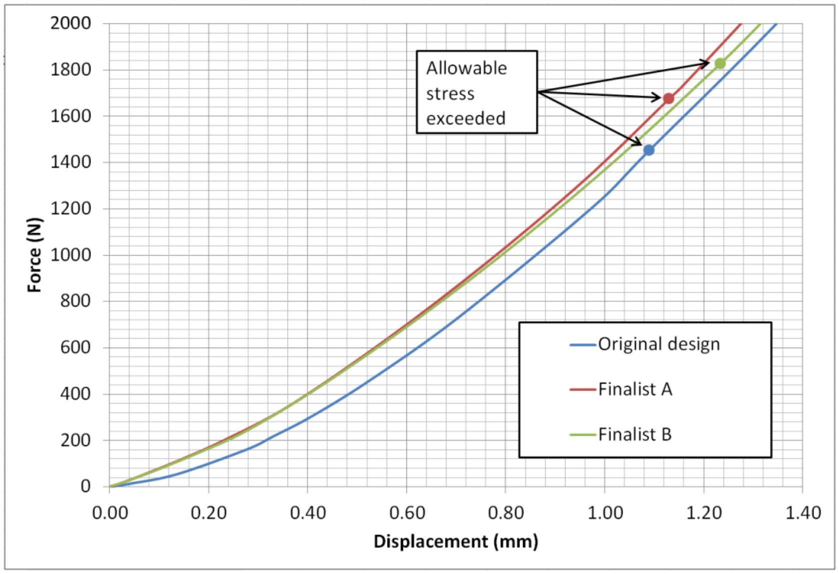

Figure 11. Force vs. Displacement for finalist designs compared to original.

In general, final verification confirmed the improvements predicted with the mass-spring analogy, while it also revealed limitations of this approach and led to the final preference for designs whose structural advantages would have been otherwise overlooked. Among these limitations are:

1. Glass Fiber Reinforced Polyproylene is an orthotropic material, with a dominant material stiffness and strength along the fibers, whose orientation is highly dependent on the mold filling pattern. This could only be predicted using the Moldflow simulation. Notice however that the material property results do not include material strength. This property is inferred from separate tests showing a connection between elastic modulus, material strength and fiber orientation.
2. This material is also found to be stronger near the surface, while it becomes weaker towards the core, due the compaction process. This again can only be predicted with the injection molding simulation. Due to this, using an average isotropic material, thick ribs would appear stronger than they really are.
3. The force-displacement response of the cleat is nonlinear. As the pole ball pushes against the socket, the contact surface tends to grow and so does the effective stiffness and strength of the cleat. So while a linear analysis can make

a good estimation of the initial trend, it cannot make an accurate prediction of the state of stress at the point where failure of the cleat would be expected.

4. Greater stiffness does not necessarily mean greater strength. As stiffening features are incorporated, the structure in general tends to become stronger, but these new features may also shift or magnify peak stress spots, where failure will eventually initiate.

4.6 Limitations of the ATOM approach

ATOM provides decisive input for the designer, who will also benefit from a major reduction in workload. However, the designer must still come up with a final functional interpretation taking into account several additional geometric restrictions such as uniform rounds, smooth parametric profiles for the ribs, and continuity of the surfaces.

Although ATOM can be applied to a nonlinear model (as long as convergence of the solution is not a recurring issue) this would have substantially increased the run-time for the solution. One potential problem is that the surface of the cleat that interacts with the base would change as a result of the optimization. This could pose a major complication for the solution of contact. Indeed one of the unexpected complications during final nonlinear evaluation of the designs was that the added ribs would interfere with the test base.

Introducing Moldflow simulation results at each iteration job within the optimization process appears impossible. The main reason is that ATOM iterations don't involve a true change in the part geometry but a change in effective material density. Moldflow would be incompatible with this approach, as it requires a new geometry for each simulation.

5. Conclusions

A procedure was devised which confirms the possibility of importing Moldflow results into Abaqus CAE in order to continue the construction of a model and add rigid parts to the assembly. This approach proved useful to obtain a reliable assessment of the structural behavior of the injection molded part and to complete the final stages of a structural optimization.

A simplified linear analysis approach emulating the results of the nonlinear analysis via a frequency analysis and a spring-mass analogy was used successfully to drive a structural optimization process. This approach proved useful to speed up a process that would have been too costly and slow with a nonlinear approach.

The Abaqus Topology Optimization Module proved to be able to find the same optimization features as in a manual approach, as well as additional features and subtleties whose exploration would have added another order of complexity in the manual approach. Furthermore, by working with the same input geometry, it relieved the designer from a considerable amount of work in generating exploratory geometry changes.

Although the results from ATOM cannot be taken as a final design solution, they do provide decisive guidelines for the designer to generate it.

Given the inherent limitations of an analogy-based linear analysis approach, a more realistic nonlinear approach considering also the effects of the injection molding process is still necessary to complete the optimization of the cleat.

6. References

1. Abaqus Users Manual, Version 6.12-1, Dassault Systémes Simulia Corp., Providence, RI.

2. Sundararajan, C., "Compendium of Formulas for the Structural Vibration Frequency Analysis of Beams," Course No. S-3001, PDH Engineer. Houston, Texas, 2009.

3. Young, W. C., and Budynas, R. G., "Roark's Formulas for Stress and Strain," McGraw-Hill, Seventh Edition, 2002.

7. Appendix

7.1 Normal workflow for an Abaqus model based on Moldflow results

The following is a typical workflow using a three-dimensional solid Moldflow simulation, as provided in the Abaqus Documentation:

1. Run a Moldflow simulation
2. Run the Visual Basic script `mpi2abq.vbs` to export the data to a file named a *job-name*`_mesh.inp` file and the result data to various `.xml` files.
3. Run the Abaqus Interface for Moldflow to create a partial Abaqus input file from the Moldflow interface files. In the present case the command used was the following:

   `abaqus moldflow job=`*newname* `input=`*job-name_mesh* `3D element_order=2 initial_stress=on`
4. Edit the Abaqus input file to add appropriate data for the analysis.
5. Submit the Abaqus input file for analysis.

7.2 Workflow for a Moldflow-Abaqus model involving an assembly

This procedure overcomes limitations that are found when trying to incorporate a part imported from Moldflow output into an assembly containing rigid parts:

1. Open the *newname* input file created by Abaqus Interface for Moldflow with a text editor. Save a copy of the original as *newname-IC.inp*. Delete the `*INITIAL CONDITIONS, TYPE=STRESS` command and data. Save as *newname-NIC.inp.*
2. Import *newname-NIC.inp* into a CAE model.
3. Set Model Attributes in CAE to "Do not use parts and assemblies in input files".
4. Add only rigid parts to the assembly besides the deformable cleat. Additional deformable parts can generate additional conflicts with material properties imported from the Moldflow results.

5. Leave the imported step to allow relief of initial stress. Provisional boundary conditions as imported may need to be modified or replaced with soft springs.
6. Add other steps, interactions and boundary conditions as needed.
7. Use Mesh/Global numbering control to make sure there will be no conflicts in node or element numbering when generating the input file from Abaqus CAE.
8. Create a job with the same name as the original Abaqus Moldflow job (*newname*).
9. Generate the *newname* input file from the Abaqus CAE job. Notice it may overwrite the existing *newname.inp*. Make sure a copy of the original (*newname-IC.inp*) was previously saved.
10. Open the newly created *newname* input file with a text editor.
11. Under the commands `*ORIENTATION` and `*SOLID SECTION`, change the orientation name to match the original (*`ABQMPI3D`*).
12. In the `*ELSET` and `*SOLID` sections, change the element set name to match the original (*`EALL`*).
13. Copy the command `*INITIAL CONDITIONS, TYPE=STRESS` and its data from *newname-IC.inp* and paste it after the after the *SOLID SECTION command.
14. Save *newname.inp* and execute it from the Abaqus command.

8. Acknowledgements

Sean Dreyer from Action Plastics in Christchurch, New Zealand, performed the solid modeling for the cleat design and generated all the design changes for analysis.

Keith Hague from Imtech Design Ltd, in Melbourne, Australia, performed the injection molding simulations using Moldflow and produced the input and data files for the Abaqus Interface for Moldflow.

Dr. Don Campbell and Dr. James Hamilton from Matrix Applied Computing in Auckland, New Zealand, performed a proof reading of the report which led to major improvements in structure and relevant content.