Topology Optimization of an Injection Moldable Prosthetic Knee Joint

Thomas A. Reist[1], Jan Andrysek[2] and William L. Cleghorn[3]

[1]University of Toronto, tom.reist@utoronto.ca
[2]Bloorview Research Institute, jandrysek@bloorview.ca
[3]University of Toronto, cleghrn@mie.utoronto.ca

ABSTRACT

One of the major challenges associated with the development of prosthetic knee joint components for individuals with above-knee amputations relates to the need to design light-weight structures that are capable of withstanding the high loads which are typically present during mobility. This work investigates the application of topology optimization to prosthetic design, and details the structural optimization of a new prosthetic knee joint that is currently under development. Optimization was performed using Altair's OptiStruct software utilizing the topology optimization method. The design was optimized in accordance with ISO-10328 specifications for structural testing of lower-limb prostheses. Using both structural integrity and injection moldability constraints a material reduction of 15.6% was achieved along with a decrease in overall deflection and a final design suitable for injection molding.

Keywords: topology optimization, prosthetic knee, OptiStruct, ISO structural testing.
DOI: 10.3722/cadaps.2010.247-256

1 INTRODUCTION

Optimization of structural aspects of a prosthetic knee joint for above-knee amputees can help to facilitate the development of prosthetic components that are lighter and more compact, thus improving their overall utility as well as performance. Moreover, achieving strength through geometry optimization rather than the use of high-strength materials, such as aerospace alloys as is conventionally done in prosthetics, makes it possible to develop more affordable prosthetic components. Most existing lower limb prosthetic knee joints cost $1,000s making them inaccessible for many individuals, especially those residing in countries with limited health care resources. Some of the keys to making these prosthetics affordable include making them from less expensive materials such as plastics instead of alloys, choosing less expensive manufacturing methods such as injection molding instead of machining and the application of mass production. However, these prosthetic components must continue to be capable of withstanding the high loads that are typically present during mobility, and provide adequate strength in accordance with existing international standards.

Topology optimization has gained considerable attention in recent years as a useful technique for reducing mass and improving the performance of mechanical parts. Certain industries, particularly automotive and aerospace, have devoted substantial resources to topology optimization due to its potential to yield significant cost savings and/or performance improvements. An example of this is the use of Altair's OptiStruct in the optimization of the Airbus A380's leading edge wing ribs and fuselage door intercostals which yielded an estimated weight savings of 1000kg per aircraft [4]. To demonstrate the application of topology optimization in prosthetic design, the optimization of a prosthetic knee joint that is currently under development at Bloorview Kids Rehab (BKR), was performed. The existing design, although structurally sound, was postulated to use excess material, thus resulting in an unnecessarily heavy and costly part.

Thus the objectives of this work were to:

- Investigate the possible applications of modern topology optimization software in the design of prosthetic components
- Through material reduction, minimize the mass of an existing prosthetic knee joint prototype for both greater user comfort and overall cost savings
- Achieve an injection moldable design to facilitate less costly production

The structural testing criteria for lower limb prosthetics is outlined in the ISO-10328 A100 specifications. For this particular prosthetic design there are three relevant test criteria. The first two (LCI and LCII) simulate the two most extreme loading conditions during the gait cycle. These occur shortly after heel-strike during load response when the prosthesis accepts the user's full weight (see Fig. 1(a)) and in late stance just prior to toe-off, as the user propels himself forward on the ball of the foot (see Fig. 1(b)). The third loading condition (KL) tests the locking mechanism (if present in a knee) and is shown in Fig. 1c. Such a mechanism ensures that the knee will not inadvertently bend when the user loads the limb. Tab. 1 shows the magnitudes of the three load conditions specified by ISO-10328.

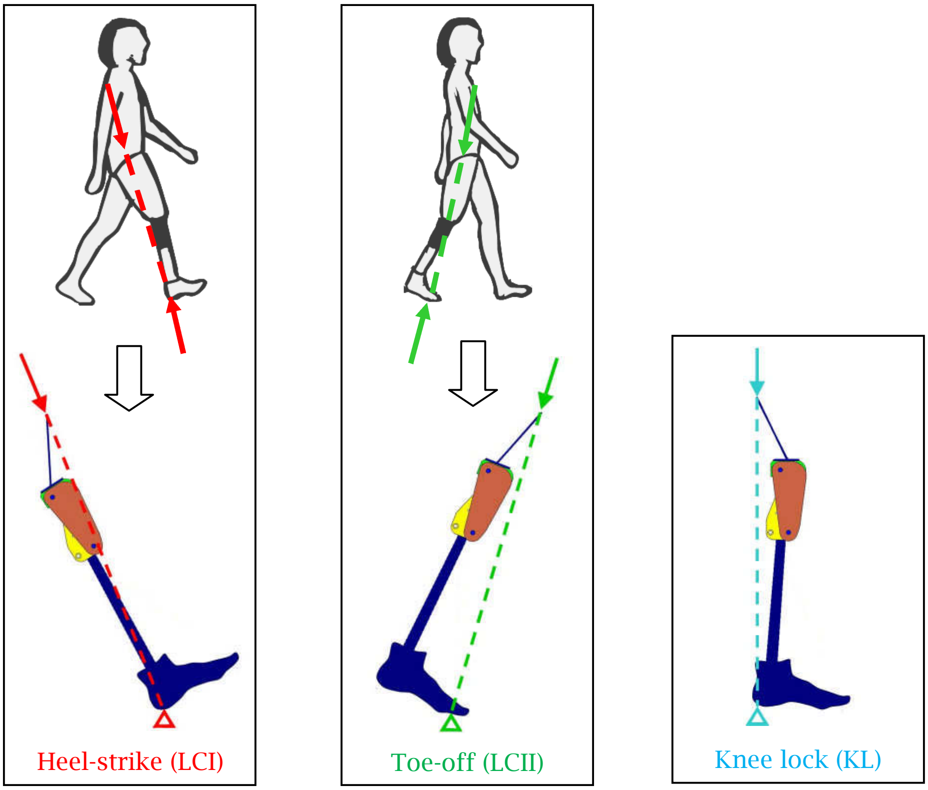

Fig. 1: Gait cycle loads and their application as per ISO-10328 specifications: (a) Load application after heel-strike (LCI), (b) Load application at toe-off (LCII), and (c) Knee lock test loads.

Load Conditions	*Magnitude* (N)
Heel-strike (LCI)	4480
Toe-off (LCII)	4025
Knee lock test (KL)	3500

Tab. 1: ISO-10328 loading condition magnitudes.

While a component of each load acts in a plane as shown in Fig. 1, the total resultant load occurs in three dimensions and passes through the foot and the center of the hip, thus creating offset loads, as would be expected, and as shown in Fig. 2.

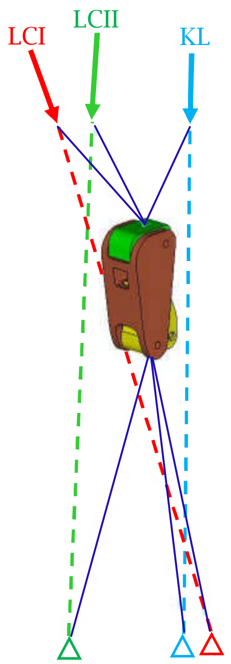

Fig. 2: Application of test loads applied to the FEA mode, shown in three dimensions.

2 METHODOLOGY

The FEA suite HyperWorks 9.0 by Altair Engineering Inc. was used for the optimization. The suite contains HyperMesh as the pre-processor, RADIOSS as the finite element analysis (FEA) solver, OptiStruct as the optimizer and HyperView as the post-processor.

2.1 Pre-processing

The CAD geometry of the original prosthetic knee was obtained from BKR and is shown in Fig. 3. This needed to be modified to remove such features as fillets and other excessive details to facilitate meshing.

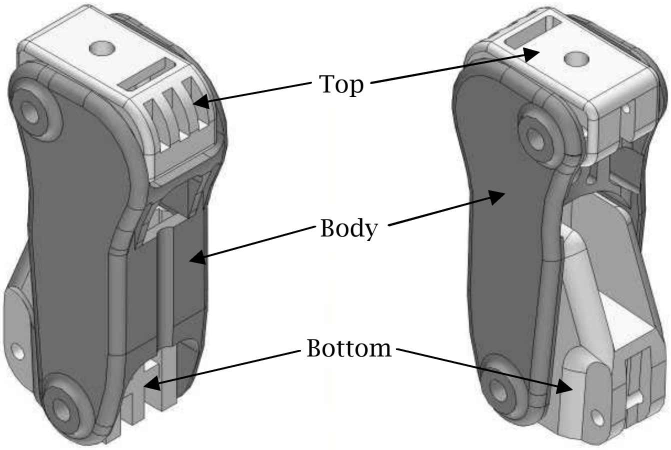

Fig. 3: The original prosthetic geometry showing both front and rear angle views.

From this geometry a design space was created. This design space was obtained by expanding the existing geometry to create a very simplified envelope within which the final design must reside. This design envelope is determined by the size and operation constraints. The design space for each part (top, body and bottom) is shown in Fig. 4 after meshing.

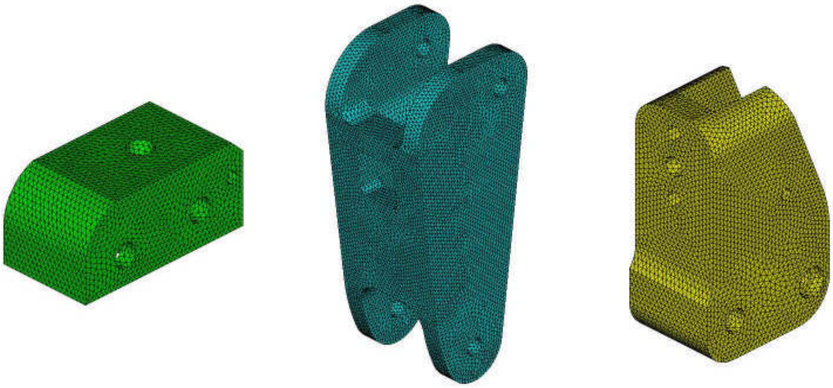

Fig. 4: The optimization design space for each part: (a) Top, (b) Body, and (c) Bottom.

The first stage in pre-processing was the definition of materials (Delrin® for the three prosthetic components, and steel for other components such as bolts, testing pylons, etc. [3]) and element properties in HyperMesh. Once these were created with the appropriate qualities the model was meshed. A 2D membrane mesh was created for the surface of each part as this allows better control of mesh quality and is required when using gap elements to model contacts. Once this mesh is created it can be 'tetrameshed' to create a solid 3D mesh. An average element size of 2mm was chosen for the design space as this allowed for the application of minimum and maximum member size constraints which constrained the wall thickness of the optimized part to be between 4 and 7mm which is a permissible range for injection molding, thus yielding a moldable design. Note that mesh refinement was not used around features such as holes, since for topology optimization a finer mesh was used over the whole part to produce more discrete results, and high accuracy at critical locations was not as critical as when performing analysis.

For optimization three parameters need to be specified during pre-processing: design variables, objectives and constraints. For topology optimization the design variable is the relative element density (see Section 2.2). For this optimization several objective/constraint combinations were considered. Due to the nature of topology optimization methods, the use of volume fraction and compliance, defined as

$$C = \frac{1}{2} u^T K u = \frac{1}{2} \int \sigma \epsilon \, dV \, , \tag{2.1}$$

where u is the deformation and K is the stiffness matrix, are well suited as objectives and/or constraints [1]. Performing FEA on the original prosthetic knee (as shown in Section 2.2) allowed the determination of the allowable compliance of the prosthetic. Since it is known that the original prosthetic meets operational requirements, the optimization can aim to achieve these same performance measures while using less material. This method is suggested in [4] based on a survey of various combinations of compliance/volume fraction objective/constraint methods. The authors found that the most favourable objective/constraint formulation when dealing with compliance and volume fraction is, if an existing design is available, to use the performance of this as a constraint function and have a minimum volume fraction objective. Another promising option was the use of a stress constraint. While historically there have been issues with using stress constraints in topology optimization [1, 2], it was found that OptiStruct handled these constraints well, and they have been successfully used by others [6]. Other constraints were examined such as deflection constraints, but due to poor definition for this problem they were eliminated in favour of the above mentioned compliance and stress methods. Following the method suggested in [4] the compliance of the model was constrained to meet that of the original design, with an additional stress constraint. The objective was to minimize the design volume fraction. To ensure a realistic as well as a moldable part, moldable member size, draw direction and symmetry constraints were also imposed. Thus, the optimization problem was defined with the above loading conditions, constraints and objectives.

2.2 Processing

Before the optimization was performed, an analysis was performed on the original part in order to have a performance reference for the optimized design (pre-processing was similar to that mentioned above but with the original geometry and no optimization parameters.) This was required to determine the compliance of the original prosthetic for use as a constraint function.

OptiStruct solves the structural optimization problem given by:

$$\begin{aligned} &\min \ f(x) = f\{x_1, x_2, \ldots, x_n\} \\ &\text{subject to:} \quad g_j(x) \le 0 \qquad j = 1, \ldots m \\ &\qquad\qquad\quad x_i^L \le x_i \le x_i^U \qquad i = 1, \ldots n \end{aligned} \tag{2.2}$$

Where $f(x)$ is the objective function, $g(x)$ is a constraint function and x_i is a design variable. OptiStruct's topology optimization uses the Solid Isotropic Material with Penalization (SIMP) element density method, which varies the relative density ρ between 0 and 1, which affects the elasticity tensor of the element given by

$$\frac{E}{E_0} = \left(\frac{\rho}{\rho_0} \right)^p \tag{2.3}$$

where E is the elastic modulus of the element, ρ is the relative element density and p is a penalization factor, chosen for this analysis to be 5 based on the relation

$$p \ge \max \left\{ 15 \frac{1-\nu_0}{7-5\nu_0}, \frac{3}{2} \frac{1-\nu_0}{1-2\nu_0} \right\} \tag{2.4}$$

given in [1] where ν_0 is Poisson's ratio of the material. The above relation yields a minimum penalization factor of ~3 which was increased slightly to promote the discreteness of the results. Once the optimization was completed, post-processing was performed as described in Section 2.3.

2.3 Post-processing

The most involved part of the optimization process was post-processing. The optimization results from OptiStruct are best viewed as element density iso-plots. These show the elements in the design space at a certain threshold density. The optimization objective (volume fraction) converged to a value of ~60%, thus this was taken as the cut-off threshold for the iso-plots. Fig. 5 shows the resulting iso-plots for the three parts.

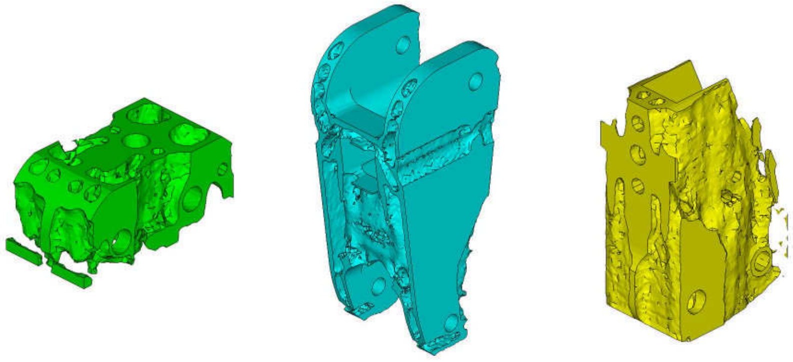

Fig. 5: Optimization element density plots at 60% threshold for: (a) Top, (b) Body, and (c) Bottom.

These iso-plots were exported into SolidWorks and discrete CAD geometry was created by manually 'tracing' features over the iso-plot surfaces. A variety of sample steps are shown in Fig. 6 with the CAD geometry in light grey and the iso-plot surfaces in dark grey. This process is quite subjective, and a different geometry could result based on the user. The final CAD geometry is shown in Fig. 7.

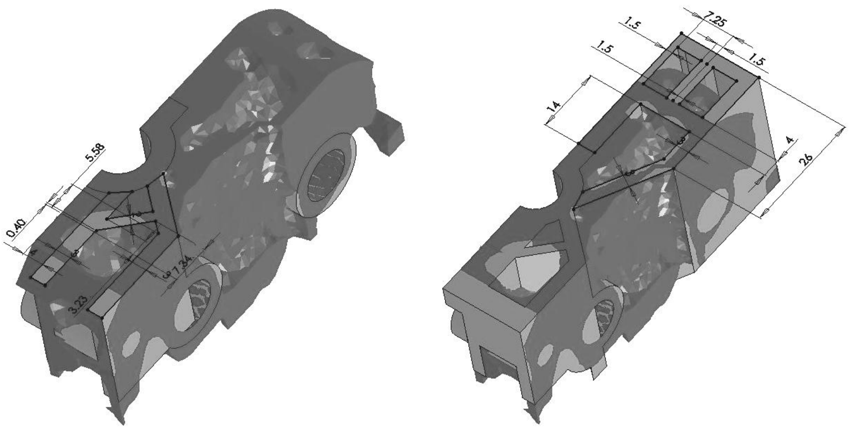

Fig. 6: A sample showing how features were sketched on the iso-plot surfaces to create discrete geometry: (a) A sketch and extrusion to create the rear ribs, and (b) A sketch and extrusion for some of the front ribs.

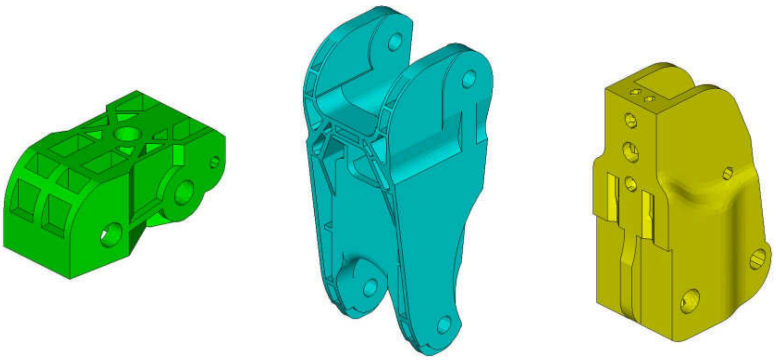

Fig. 7: Simplified optimized geometry in CAD format: (a) Top, (b) Body, and (c) Bottom.

Once this design was achieved, the geometry was brought back to HyperMesh and FEA was performed to determine how it compared to the original parts. Based on these results slight modifications were made to the optimized parts to reduce stress in certain areas. Further slight modifications were made to try to make the parts as injection moldable as possible. This included providing uniform wall thicknesses, sufficient corner radii and draw direction clearances. Fig. 8 shows the final optimized knee assembly.

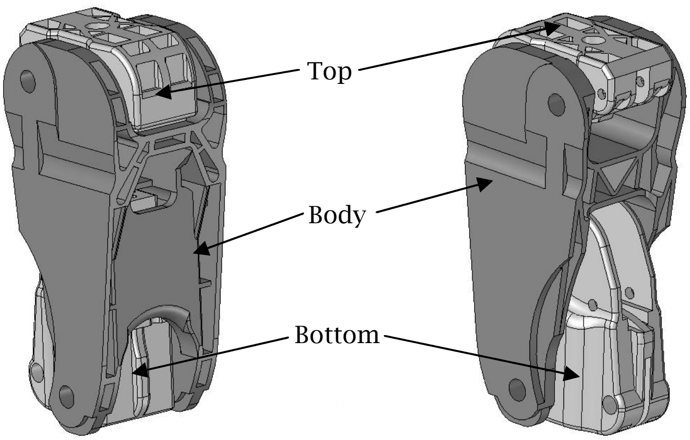

Fig. 8: The final optimized knee geometry showing both front and rear angle views.

3 RESULTS

The results of this optimization can be viewed through several measures. Firstly, an overall reduction in mass of 15.6% was achieved through this optimization as shown in Fig. 9.

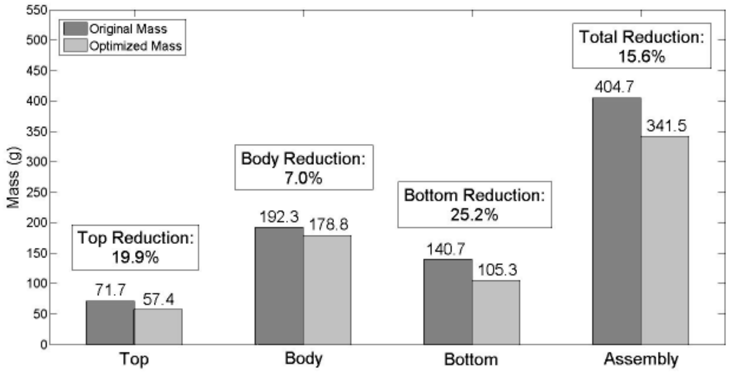

Fig. 9: The mass reduction of each part and the overall prosthetic achieved through optimization.

Secondly, the compliance for each load condition decreased. A lower compliance indicates a decrease in the structure's strain energy and an increase in its stiffness, which is desirable so long as the structure is strong enough to support this increase in rigidity. Fig. 10 shows the compliance under the three load conditions for both the original and optimized design. From this it can be seen that main reduction in compliance came from load case LCII. This is due to the fact that LCII exhibits the most eccentric load and thus causes a significant amount of body twist about the vertical axis. The optimized part has a more splayed stance throughout the body (while remaining inside an envelope the same size as the original). This allowed it to more efficiently resist the twisting due to LCII. Overall the structure's weighted compliance decreased by 69.4%.

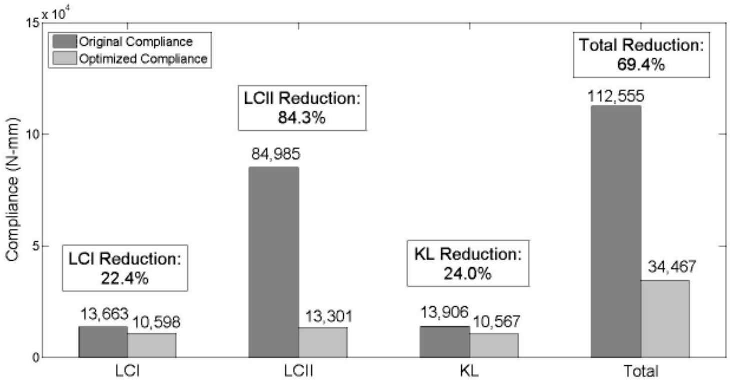

Fig. 10: The compliance for each load condition and total compliance achieved through optimization.

The effects of this lowered compliance can be seen in the deflection data shown in Fig. 11. For all three load cases a reduction in deflection magnitude was achieved. It can be noted that the reduction in deflection is not proportional to the compliance reduction when looking at LCII. This is due to the large amount of rotational deformation in the original part under LCII as described above, which is not present in the optimized part. The linear deformation (shown here) is not reduced as much as the rotational deformation.

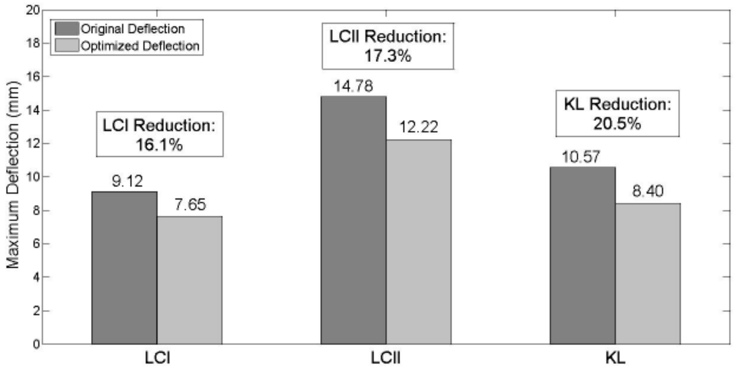

Fig. 11: The maximum deflection for each load condition achieved through optimization.

The optimized design has a higher average stress distribution, which is desirable so as to use the material more efficiently. The magnitude of the maximum stress is similar in the original and optimized design and occurs at the pin holes due to the bearing loads from the connecting pins in both the original and optimized parts.

4 CONCLUSION

Topology optimization is a robust optimization technique which has seen widespread use in recent years in particular industries such as automotive and aerospace. This project has shown that the same methods can be applied to prosthetic design and optimization. Using this method in accordance with ISO-10328 test specifications an existing design has been optimized to reduce material usage by 15.6%, reduce the structure's compliance by 69.4%, reduce the deflection for each load case by at least 16%, as well as produce a design suitable for injection molding.

The optimized design has been arrived at numerically through this work, but as part of future work physical verification is needed. The FEA results for the original prosthetic knee agree well with the results of previous structural tests, by suggesting failure in the same location as was observed during testing. This lends credibility to the results obtained herein; however, structural testing should always be performed to verify numerical results. The design will first need to be reviewed by an injection molding consultant for moldability and then tested. This will determine the validity of the optimized design.

ACKNOWLEDGMENTS

The authors would like to thank Mr. A. Furse for his assistance with the background information and original design work. The authors would also like to thank the National Science and Engineering Research Council of Canada of Canada (NSERC) for their financial support which made this work possible and Altair Engineering Inc. for the generous contribution of their HyperWorks suite.

REFERENCES

[1] Bendsoe, M. P.; Sigmund, O.: Topology Optimization: Theory, Methods and Applications, Springer-Verlag, Berlin, Germany, 2003.
[2] Cheng, G.; Jiang, Z.: Study on Topology Optimization with Stress Constraints, Engineering Optimization, 20(2), 1992, 129-148.
[3] Dupont™ Delrin® 127UV BK701 Acetal Resin Product Information, www.dupont.com.
[4] Krog, L.; Tucker, A.; Kemp, M.; Boyd, R.: Topology Optimization of Aircraft Wing Box Ribs, 10th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference, AIAA-2004-4481, Albany, New York, 2004.

[5] RADIOSS/OptiStruct User's Guide, Altair Engineering Inc., Troy, MI, 2008, www.altairhyperworks.com.
[6] Rao, J. S.: Recent Advances in Optimization of Aerospace Structures and Engines, Springer Proceedings in Physics, 126, 2007, 323-333.