Case Study

Simulation-driven design for a Joint Replacement with Ansys Discovery

Developed and curated by the Ansys Academic Development Team

Lakshana Mohee and David Mercier

education@ansys.com

Summary

This case study showcases how Ansys Discovery can be used to design a hip joint implant for simple loading conditions. It demonstrates how live simulation-driven design can be used to optimize the hip joint implant material and evaluate trade-offs between different geometries. It is a continuation of the case study titled "Level 3 Industrial Case Study: Biomaterials Selection for a Joint Replacement" where the top materials candidates for a total hip replacement are identified using the Ashby Advanced Materials Selection Methodology.

Table of Contents

1. Problem Statement 3
2. Material and Geometry Definition 4
3. Pre-processing 5
4. Solution Method, Post-processing, and Optimization 6
5. Trade-off Analysis 7
6. Possible further Analyses 9
7. Conclusions 10

1. Problem Statement

In silico testing in healthcare is becoming increasingly cost-effective and time saving to understand the behavior of medical devices before conducting clinical trials. It also enables regulatory approval while prioritizing patient safety. The objective of this case study is the development of an innovative modeling framework based on the coupling of analytical material selection methodology with a simulation driven design approach (see Figure 1), that supports material and geometry optimization for new medical device development. A material selection project carried out with Ansys Granta EduPack software, which contains advanced bio-engineering materials databases, found the best material candidates that have met design constraints and objectives of a total hip joint replacement. In this case study, a computer-aided design (CAD) model of a hip implant is imported into Ansys Discovery where it is simulated using finite-element analysis (FEA). The model is subjected to a structural force that is a simplified version of the real-life scenario of a hip implant (see Figure 2). The software facilitates meshing of the model and delivers instant multiphysics simulation. An instantaneous visualization of stress and strain fields into the hip joint led to an optimized combination of material and geometry, resulting in relevant new designs of a common hip joint.

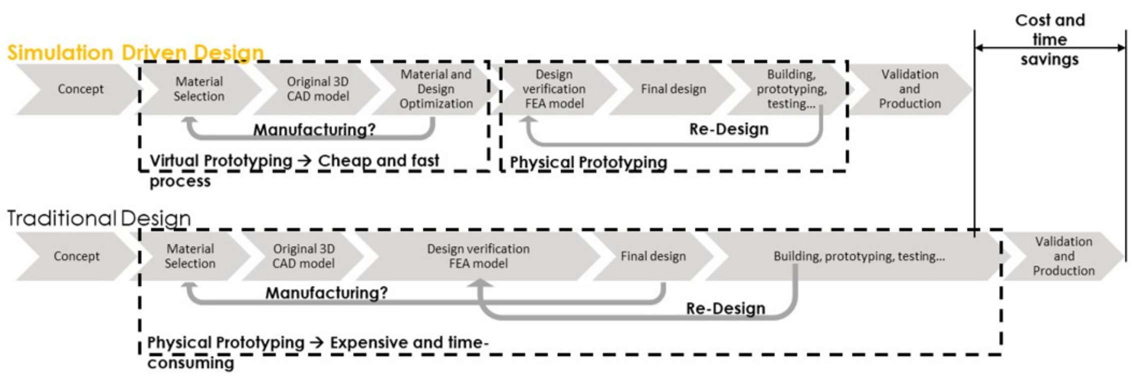

Figure 1. Flowcharts of material and design optimization in the case of simulation driven design (top) and traditional design (bottom).

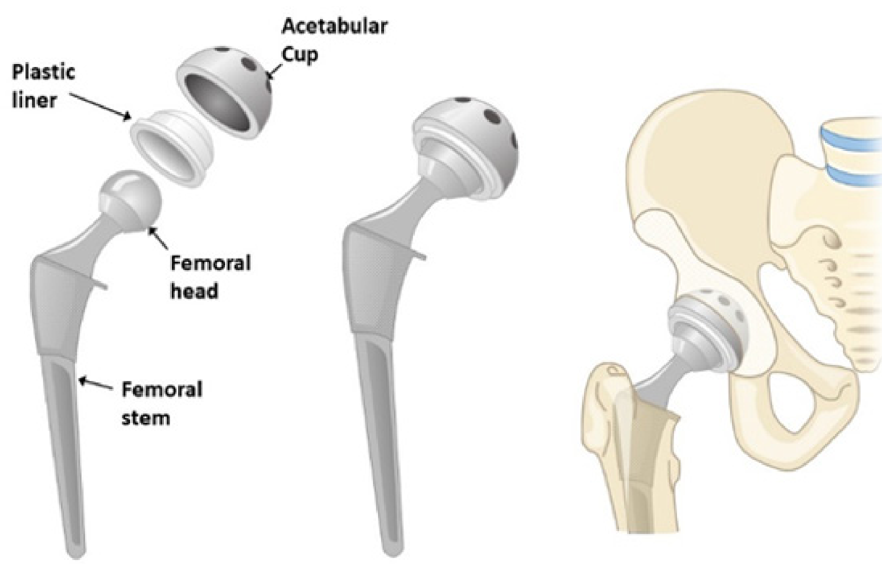

Figure 2. Description of the different part of a hip joint (on the left) and the human bones integration (on the right).

2. Material and Geometry Definition

A material selection project, carried out with Ansys Granta EduPack, found that the best materials for the stem of a hip joint are:

- Titanium alpha-beta alloy, Ti-6Al-4V aged
- Stainless steel, austenitic, BioDur 108, 30%-40% cold worked
- Nickel-Chromium alloy, solution treated and aged
- Cobalt-base-superalloy, CCM, warm worked (low carbon)

You can find a detailed description of the selection project at the following link: Level 3 Industrial Case Study: Biomaterials Selection for a Joint Replacement. The material datasheets were imported into Ansys Discovery from Granta EduPack (Level 3 Bioengineering – MaterialUniverse) using export/import functions embedded within the software. If the Granta EduPack module is not available, the materials or ones with comparable mechanical properties can be found in the Ansys Discovery Material Library.

These four materials all have similar mechanical behavior in terms of mass per unit of strength and mass per unit of compressive strength. They differ with regards to cost per unit of compressive strength. The titanium alloy, Ti-6Al-4V, is a good compromise between mechanical and cost performances and is used widely for such applications. Stainless steel, BioDur 108, has a lower price, making it also a good material candidate for hip implants. Nickel-chromium alloys and cobalt-base-superalloys have been linked to causing allergy reactions, according to ASM Medical Materials Database™. The next step in the workflow is to model and to optimize the hip implant computationally and to study the effect of the material choices on the mechanical behavior.

	Titanium, alpha-beta alloy, Ti-6Al-4V, aged	Stainless steel, austenitic, BioDur 108, 30-40% cold worked	Nickel-chromium alloy, INCONEL 718, solution treated & aged	Cobalt-base-superalloy, CCM, warm worked (low carbon)
Computed Properties				
Mass per unit of strength	41.5 - 43.7	55.4 - 69.2 ⇧	76.9 - 82.2 ⇧	79.2 - 92.5 ⇧
Mass per unit of compressive strength	3.85 - 4.02	4.71 - 6.59 ⇧	7.15 - 8.83 ⇧	7.75 - 9.76 ⇧
Cost per unit of compressive strength	71.1 - 79.6	6.71 - 9.58 ⇩	83.4 - 106 ⇧	161 - 222 ⇧

Figure 3. Comparison table from Ansys Granta EduPack; giving the increase or the loss for each material performance indices.

Two simplified CAD models of hip joint (stem + head) are evaluated. The key distinction between the two otherwise identical models are the holes found on the stem; presented in Figure 4. The radius of the head is 20mm, the overall length of the stem is about 120mm and the circular top part of the stem has a radius of 16mm. The radius and the height of the smaller cylinder between the spherical part of the head and the stem, are respectively 7.5mm and 10mm. The inclination between the stem and the head is about 40°.

The initial geometry is presented in the figure below and can be also found in the case study folder under the name HipJoint_Initial_Geometry.scdoc.

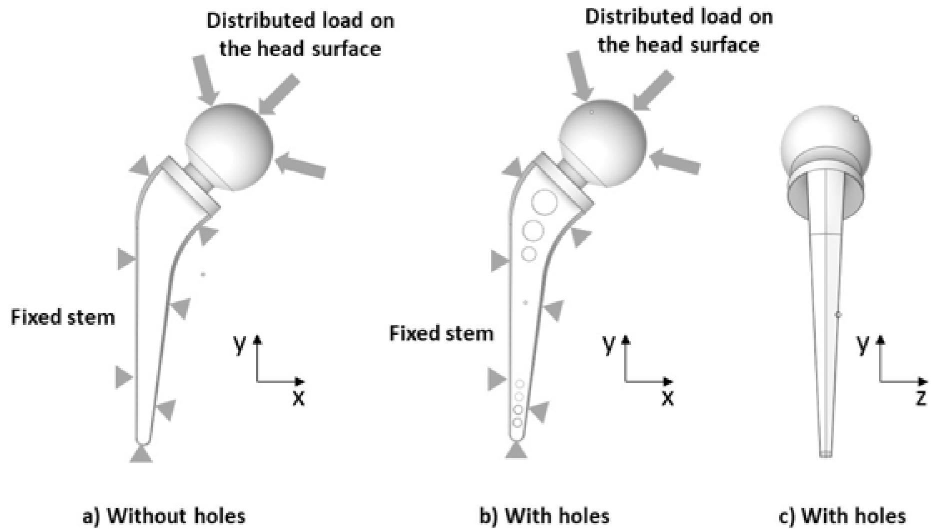

Figure 4. Views of the hip joint CAD model: a) without holes (y- and x-axis plane), b) with holes in the stem part (y- and x-axis plane) and c) for both design(y- and z-axis plane) without BCs.

3. Pre-processing

In this section, the simulation set-up to solve the hip joint replacement will be discussed in detail. To understand more about Ansys Discovery, it is recommended to check the Ansys Discovery Forum to learn the basics of the software.

All the different bodies (stem, head and the connection between the stem and the head) are all inter-connected without any allowed movement (neither interpenetration nor gliding motion).

A homogeneous distributed load of about 3 kN has been applied on the femoral head surface and the stem has been constrained as a fixed support (as shown in Figure 5). The applied load of 3 kN was estimated from the weight of an average adult of 80 kg during a standard displacement (such as no jumping) according to Paul, John P. (1967)[1]. The whole model was assigned one material candidate at a time. The temperature during analysis is set to 38°C which is the average temperature of a human body.

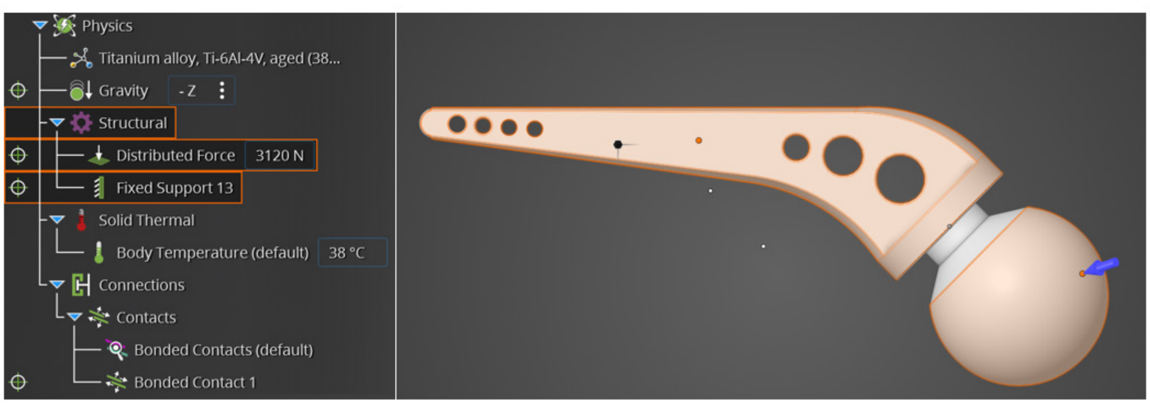

Figure 5: Simulation set-up on Discovery

1 Paul, John P. (1967). Forces transmitted by joints in the human body: SAGE, Inst Mech Eng 1967; 181: 8–15.

According to the design strategy based on a simulation driven design approach (highlighted in Figure 2), the different static structural linear analysis with thermal stress (body temperature considered for analysis) of the hip joint loading was performed using the 'Explore Mode' of Ansys Discovery (with a fidelity level of 7.7e-4m). The 'Explore Mode' is a graphics processing unit (GPU)-powered mode, which is a promising approach providing an innovative real-time FEA-solution to assess stresses, displacements, or other mechanical parameters (Von Mises, elastic strain…) without the need of meshing tools. The 'Explore Mode' provides a fast real-time simulation visualization. Using this mode, the factor of safety (FoS), the maximum Von Mises stress, the maximum deformation, the maximum elastic strain, and the mass were calculated for both designs, and presented in the next section. Tracking these parameters allows the user to respectively check that the implant will perform safely; to identify areas that may need reinforcement to avoid deformation damage to surrounding tissues; and finally to minimize implant weight.

4. Solution Method, Post-processing, and Optimization

An output of the simulation can be seen in Figure 6, showing an example of the Von Mises stress contours for the design with holes. The maximum Von Mises stress is noticeably higher around the top of the stem near the head, where a pressure was imposed. This first overview of the stress distribution allows an engineer or designer to visualize the weak structural areas of the tri-dimensional model and to modify the geometry in real time. Based on this first step, the different parameters (the maximum Von Mises stress, the maximum deformation, the maximum elastic strain) are calculated for each different configurations (type of design and materials) and are summarized in Figures 7 and 8. It is interesting to note that using the design with holes (and especially the one made with BioDur material) presents in general a higher factor of safety in comparison to the others, with a systematic increase of 20%. This effect might be attributed to the fact that design with multiple holes tends to uniformly distribute the stress lines over the geometry resulting in less stresses & increased strength. Based on this initial result, the design with holes is used for further analysis during the material optimization stage. The radial plot given Figure 8 is thus an easy way to compare all materials with the different normalized parameters.

Given this initial result, the design with holes is used for further analysis during the material optimization stage. The radial plot given Figure 8 is thus an easy way to compare all materials with the different normalized parameters obtained from calculations. According to the radial plot in Figure 8, although using BioDur for the hip implant provides a higher safety factor than with Ti-6Al-4V, the mass required is relatively higher. Hence making Ti-6Al-4V a better candidate and a good compromise.

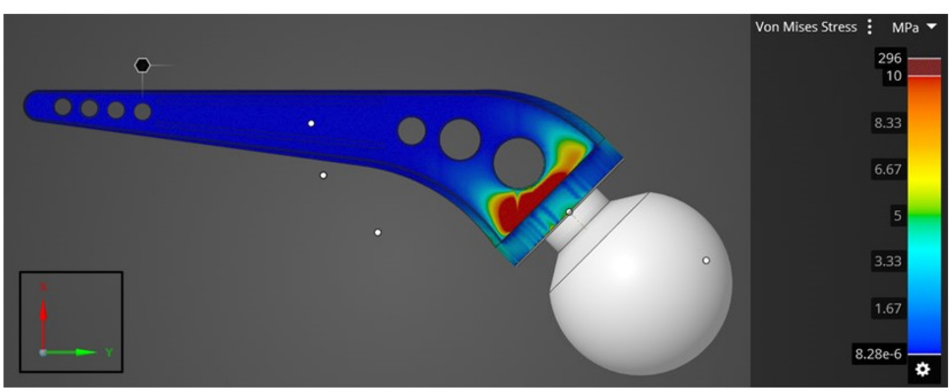

Figure 6. Simulation results in the 'Explore' mode. Von mises stress contours on the stem part only, in the case of Ti-6Al-4V. The stress concentration is around the hole just below the head part where the load is applied.

Figure 7. Evolution of factor of safety as a function of the different material and the hip implant design.

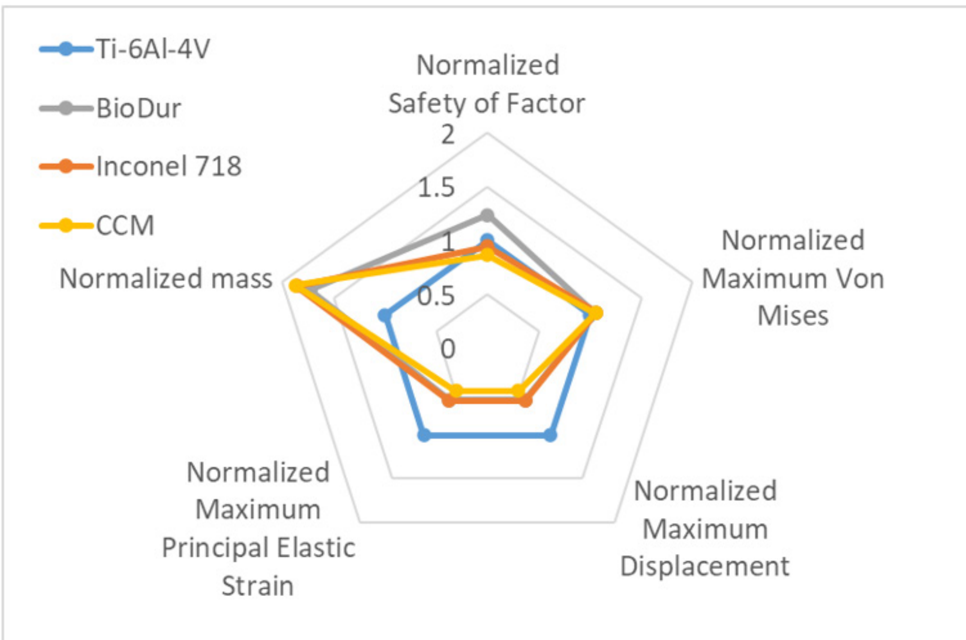

Figure 8. Radar chart to compare material performance in the case of hip joint design with holes, given different modeling results, normalized with Ti-6Al-4V values.

5. Trade-off Analysis

As a result of this investigation, we found the optimal design seems to be the hip joint with holes in Ti alloy, however the question of the actual cost of each in terms of price and environmental impact remains unknown. In Ansys Granta EduPack, in addition to engineering properties such as mechanical, thermal, and electrical, price and environmental impact information (*e.g.* carbon footprint) are also available. For the different studied materials, these amount respectively to 22.8 €/kg, 19.6 CO_2 kg/kg for Ti alloy, 1.81 €/kg, 2.64 CO_2 kg/kg for BioDur stainless steel, 15.8 €/kg, 10.5 CO_2 kg/kg for Ni-Cr alloy and 27 €/kg, 26.1 CO_2 kg/kg for Co-based alloy. As observed, these values are defined per kg of material, and to obtain the total values for each design, they can be multiplied by the weights found in Table 1, leading to the four possible optimized hip joint designs.

The question of which is the best overall design among them is inherently a difficult one and cannot be answered without answering another important question, which is how much we are willing to compromise in terms of economic and environmental cost to achieve a safer and much lighter design. A performance factor F can be established using the following general weighted function:

$$F = \sum_{i=1}^{N} C_i \frac{P_i}{\max(P_{i \to N})} - \sum_{i=1}^{n} c_i \frac{p_i}{\max(p_{i \to n})}$$

With *P* the properties to maximize of a total number of *N* (*e.g.* the safety factor), *p* the properties to minimize of a total number of *n* (*e.g.* the price, the CO_2 footprint...), *C* and *c* respectively user-defined weights to increase the importance of a property in the factor calculation taking into account for example if the designer prefers to have a better mechanical performance rather than an low environmental impact (*e.g. C>c*). Such approach can been generalized like in the paper written by Hamidi *et al.* in 2015.[2] In our case study, the previous equation can be adapted using weights of 1 for every property ($F_{CS,1}$) or user customized weights to optimize mechanical performance ($F_{CS,2}$), and obtained values are summarized in Table 1:

$$F_{CS,1} = \frac{SF}{\max(SF)} - \frac{\text{Weight}}{\max(\text{Weight})} - \frac{\text{Price}}{\max(\text{Price})} - \frac{\text{CO2 footprint}}{\max(\text{CO2 footprint})}$$

$$F_{CS,2} = 2\frac{SF}{\max(SF)} - 2\frac{\text{Weight}}{\max(\text{Weight})} - 0.5\frac{\text{Price}}{\max(\text{Price})} - \frac{\text{CO2 footprint}}{\max(\text{CO2 footprint})}$$

Table 1. Comparison of Safety Factor, Weight, Price and CO2 of the 4 material candidates.

Material	*Ti-6Al-4V*	*BioDur*	*Ni-Cr alloy*	*Co-based alloy*
Safety factor (SF)	3.54	4.40	2.45	3.06
Weight (kg)	0.09	0.16	0.17	0.17
Price (€)	2.05	0.29	2.69	4.59
Carbon footprint (kg)	1.76	0.42	1.79	4.44
Performance Factor ($F_{CS,1}$)	**0.43**	**0.90**	**-0.43**	**-1.30**
Performance Factor ($F_{CS,2}$)	**0.93**	**0.99**	**-0.58**	**-1.11**

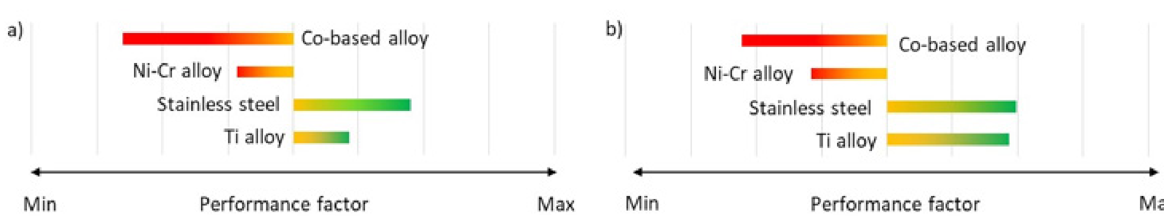

Figure 9. Comparison of Safety Factor, Weight, Price and CO2 of the 2 material candidates, a) with unitary weights and b) with user customized weights to optimize mechanical performance.

2 E. Hamidi et al., « Materials selection for hip prosthesis by the method of weighted properties", 2015. http://dx.doi.org/10.11113/jt.v75.5291

6. Possible further Analyses

The fact that stainless steel tends to be the best candidate, highlights the need to perform potentially a topological optimization (TO) or to use a lattice structure to reduce hip implant mass. Such approaches are also justified to improve such design by creating more complex shapes to reduce the prosthetic failure due to bone resorption (increase of contact surface between the bone and the implant). Such approaches can be found in the literature[3,4], coupling a biocompatible material with a complex geometry obtained by TO and built using AM techniques. On one hand, the lattice structure can be defined in Ansys Discovery using common extrusion examples, lattices or minimal surfaces designs. Examples of such design for the hip joint are given Figure 9. On the other hand, TO step requiring specific algorithms (level set method) can be performed into the 'Explore' mode of Ansys Discovery. Currently, you can use optimization to either maximize the stiffness of your topology, minimize the response to free vibration, or both. You can also target a specified frequency or optimize to remove excess material. A possible stiffness-optimized geometry for the stem of the hip implant is thus given in Figure 10. Keeping the same BCs as in the previous section and defining a protected depth near the contact between the stem and the head, the TO algorithm led to an emptied stem, keeping intact the outer surfaces of the stem.

However, for both approaches (lattice structure or TO), it is additionally required to verify the feasibility of processing such a complex geometry using Ti-6Al-4V for example, according to the workflow presented Figure 1. This can be done through a validation check using the various materials databases (Senvol, MaterialUniverse within Ansys Granta Selector), or by checking the literature. For example, manufacturing processes such as casting or additive manufacturing (AM) (Murr, Mawrence (2012)[5] and Hao, Yu-Lin (2016)[6]) demonstrate promising results to designing such complicated geometries. And to optimize the additive manufacturing of such complex shapes, it is advised to consider process simulation using Ansys Additive Suite.

Finally, as an outlook for the future of the hip joint design process, it is possible to coat the stem, with materials such as polyetheretherketone (PEEK) which have been identified as improving wear resistance (Anguiano-Sanchez, Jesica (2018)[7]). They also enhance the load transfer to the bone, minimizing stress shielding effect and thus prolonging the implant lifespan.

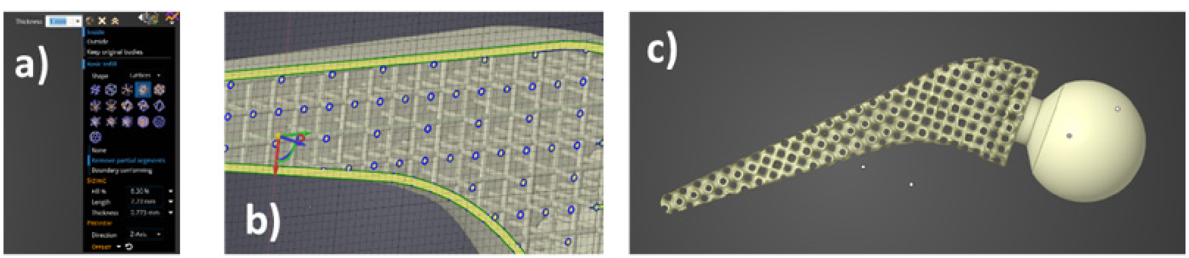

Figure 9. a) Menu to setup up an internal structure (extrusion, lattice or minimal surface) into a body using the 'shell' function. b) Zoom into an example of hip stem with a lattice structure. C) Overview of the hip joint with stem defined using a minimal surface design.

3 https://compmech.unipv.it/topology-optimization-and-additive-manufacturing-for-patient-specific-hip-prostheses/
4 H. Burton et al., "The design of additively manufactured lattices to increase the functionality of medical implants", https://doi.org/10.1016/j.msec.2018.10.052
5 Murr, Mawrence (2012). Next Generation Orthopaedic Implants by Additive Manufacturing Using Electron Beam Melting: Hindawi Publishing Corporation, International Journal of Biomaterials.
6 Hao, Yu-Lin (2016). Biomedical titanium alloys and their additive manufacturing: Springer, Rare Metals volume 35, pages 661–671.
7 Anguiano-Sanchez, Jesica (2018). Influence of PEEK Coating on Hip Implant Stress Shielding: A Finite Element Analysis: Hindawi, Computational and Mathematical Methods in Medicine.

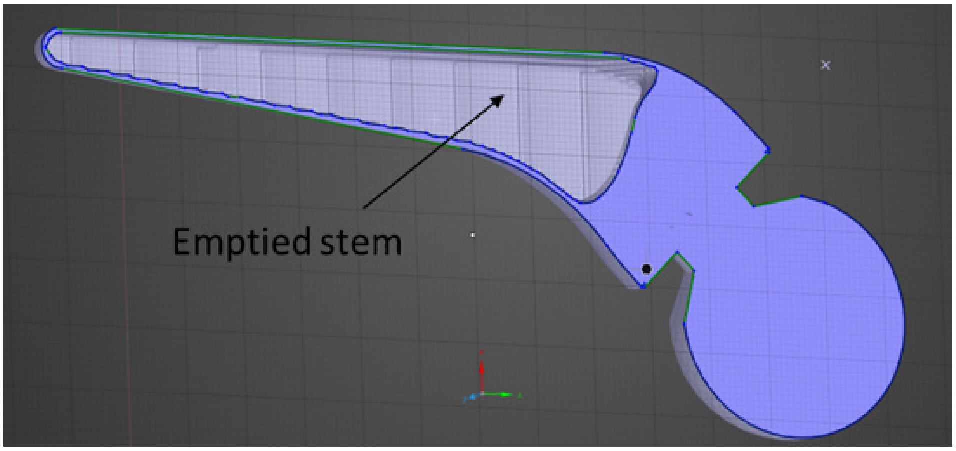

Figure 10. Topological optimized stem design with a target of 50% volume reduction.
The result is observed using the section plane tool (along the x-y plane) allowing the designer to investigate the inside of the stem body.

7. Conclusions

In conclusion, in this case study, an innovative workflow based on material selection and simulation driven design is shown and has been applied to the case of a hip joint prosthesis. Both materials and shapes for the stem part of the hip joint were simultaneously studied, simulated, and optimized by following simple and connected procedures. A combination of material + design has been thus proposed based on a trade-off analysis considering additional key properties like the cost or the carbon footprint. Finally, an optimized design has been proposed and validated from a process point of view by looping with a material database to check the technical feasibility.

Such a workflow presents numerous advantages from both a research and an industrial standpoint, by helping to save time and money in the development of a product, and by giving more optimized designs before testing via traditional numerical tools and physical prototyping. Thus, an initiative to bridge the gap between material selection and simulation has been discussed in this paper and this proposed solution might be considered in the design exploration stage to help every engineer to make better-informed decisions.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to structures, fluids, or heat transfer (physics areas supported by Ansys Discovery).

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

Acknowledgements

Dr. Nick Stefani, Dr. Janos Plocher

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com