Solution from Lattice Sizing Optimization to Additive Manufacturing

L. Cai[1], S. Wright[2], J. Fort[3], J. Miller[4], J. Bi[3]

1. Weldon School of Biomedical Engineering, Purdue University, West Lafayette, IN, US

2. nTopology inc., Brooklyn, NY, US

3. Dassault Systèmes, Johnston, RI, US

4. Mechanical Engineering Department, Purdue University, IN, US

Abstract: *Lattice structures bear many desirable characteristics from design standpoints, such as stable designs with large network of structural members, desirable weight characteristics, custom mechanical behavior and porous nature that could facilitate bone and tissue growth on medical implants. To efficiently optimize the structure of lattices, we can use Tosca structure to vary the lattices thickness to reach certain design objectives. However, the radii in Tosca are associated with beams which are not optimal to have a smooth stress transition on connected nodes. Additionally, the optimized results can not be meshed smoothly enough in Abaqus for additive manufacturing. In order to avoid these limitations, we proposed a workflow combining the Tosca structure and Element Free (nTopology) using python scripts. As a demonstration, a bike stem model was created in solid. By meshing it in Abaqus and reconstructing it in Element Free, a basic lattice structure with internal supporting beams was created and imported into Tosca for sizing optimization. Other than Tosca which has radii associated with beams, Element Free has nodal radii. To deal with this difference, the optimization results were converted into nodal radii models in Element Free. Then a STL file was created in Element free based on the converted results. Additionally, a method to design the initial RVE lattice structure using topology optimization was mentioned for those who might be interested. This whole process enabled the additive manufacture on optimized lattices and reduced the design time to one or two hours.*

Keywords: *Lattice Structure, Sizing Optimization, additive manufacturing, Abaqus, Tosca Structure, Element Free, bike stem, RVE design.*

1. Introduction

Structural optimization has been shown to be able to accelerate the design process and to save resources for various engineering disciplines. Among these optimization tasks, sizing optimization of lattices is one of the most common tasks existing in industry (Bendso, 2003). For example, lattices structures were used in aerospace which requires light weight structures (Gerzen, 2016; Vasiliev, 2012), and shoes industry where customized rubber lattices are needed, and biomedical devices which have organic internal structures of implant devices. Moreover, the importance of sizing optimization of lattice structures increased due to the constant expanding possibilities of the additive manufacturing technology (Gerzen, 2016). Most models of this type lead to large scale of structures which need powerful optimization design capabilities. Such solutions are implemented in the SIMULIA Tosca Structure. However, there was another gap between the lattice sizing optimization and additive manufacturing, which is the meshing of the optimized lattices. This was accomplished by the use of software Element Free (Element Free Manual, 2017). The design capability of Element Free can also help in the initial lattice creation to reduce the requirements for support structures in additive manufacturing.

2. Methods

In the present study, lattice sizing optimization was demonstrated in a workflow shown below (Figure 1). It was mainly composed of three parts: (1) Lattice construction from a solid model (Abaqus Manual, 2016). (2) Sizing Optimization in Tosca Structure (Tosca Manual, 2016) (3) Lattice meshing in Element Free (Element Free Manual, 2017). The conversion of models between Abaqus and Element Free was accomplished using Python scripts. The files in Element Free can be exported to be an easy interpreted LTCX file, which is an open source XML format. It was to note that Element Free has beam radii associated with nodes, creating a lot of tapered beams which are not compatible with Tosca. To convert between tapered beams and constant radii beams, an interpolation method was implemented and a conversion accuracy test was done.

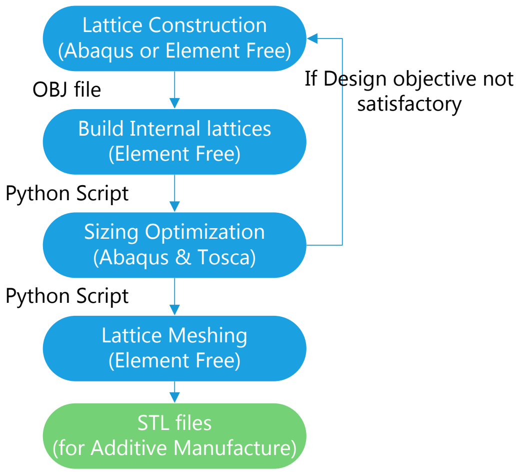

Figure 1. Work flow of lattice sizing optimization and lattice meshing.

3. Results

3.1 Lattice Construction

Lattice construction was completed from surface mesh of a solid model in Abaqus. Once the solid model was meshed either in hexahedral or tetrahedral elements, the surface meshes can be converted into lattice structures. In Figure 2, a solid bike stem model was meshed and its surface mesh was converted using Abaqus plugin called “Beam elements from solid element edges”. Afterwards, the model was transferred to Element Free through OBJ file format.

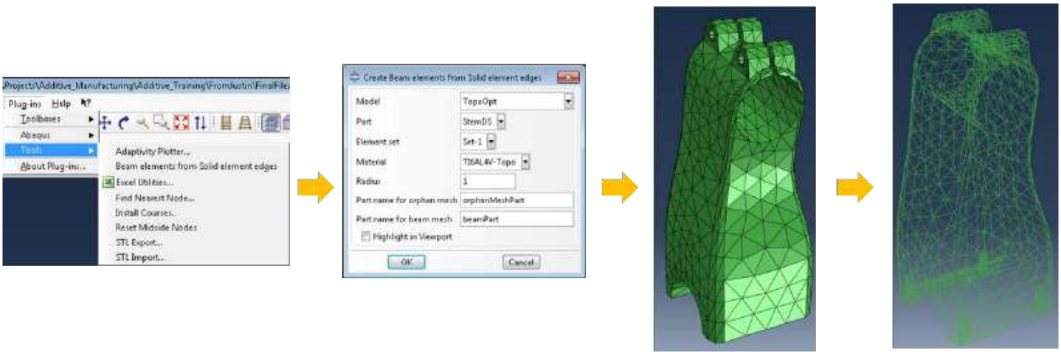

Figure 2. Lattice construction using an Abaqus plugin.

3.2 Build internal lattices

The surface lattice creation can also be completed in Element Free (Figure 3). Moreover, Element Free can produce internal lattices whose types, density and orientation can be customized to reduce supported material. Moreover, lattices within certain angle with the horizontal surfaces can be removed if too much support materials are needed. Furthermore, extra support structures can be added, such as a 45 degrees diamond shape struts in Figure 3, to reduce support materials. To customize the beam thickness, Element Free can control the average beam thickness as well as the beam thickness distribution. Once the lattice model was created, it can be exported into a light weight LTCX file, which is based on an open source XML format. With a python script, it was converted into an INP file for Abaqus.

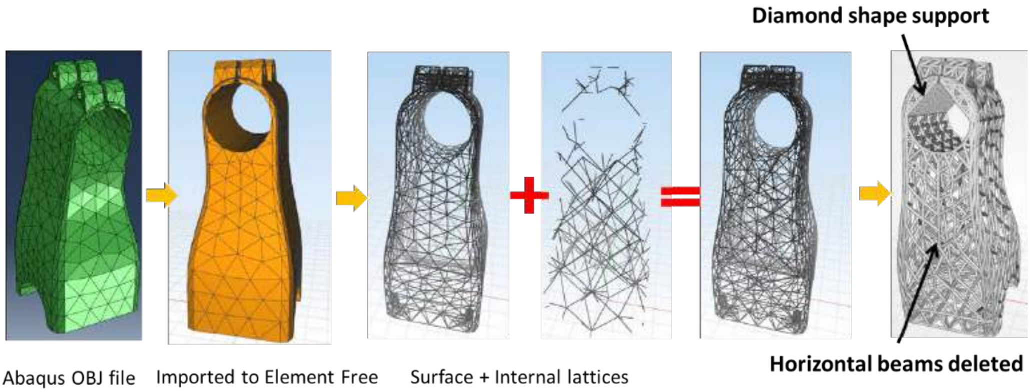

Figure 3. Alternative way to create surface lattices in Element Free and the internal lattices binding and options to reduce support material.

3.3 Sizing Optimization and lattice meshing

Once the modified lattice structure was imported in Abaqus, four different load cases were defined (Figure 4A), according to the testing standard (ISO 4210-5, 2014). In this application, the volume was constrained within four times the initial value and the maximum value of compliance was minimized (MINMAX) as an objective function. The Objective function was shown in Figure 4B as decreasing along iterations, which means the stiffness of the structure was increasing 37.8 times while volume only went up four times. Afterwards, the optimized lattice structures were converted to LTCX file for Element Free using python script. In Figure 4C, two different optimization results were derived in which the second design has most of the horizontal lattices removed. In both designs, thick "X" shape supports are visible and represent an economic way to be strong in two orthogonal loading, which include forward bending and lateral bending. It is to note that a thick "X" structure appeared on the surfaces in the first design, while the removal of horizontal lattices varied the structures in the second design. At last, the tapered lattices were meshed into STL files in Element Free, where smooth transition at connections was preserved and the stress concentration was minimized. The meshing process created millions of elements within several minutes.

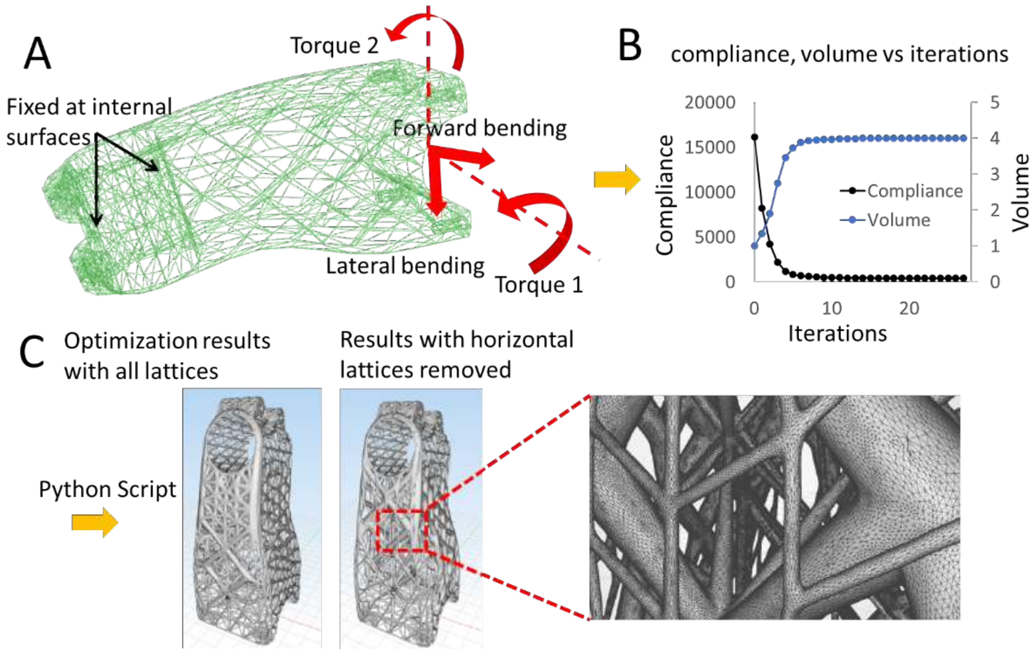

Figure 4. (A) Four loading cases were defined in Abaqus for sizing optimization (B) Objective function of strain energy was decreasing with increasing iterations (C) Optimization results with all lattices and with horizontal lattices (<30⁝ with horizontal plane) removed. Mesh details of the lattice structure in Element Free was shown on the right.

3.4 Conversion accuracy test on Python script

As described in the methods section, the beams in Element Free have radii associated with nodes to make the mesh transition smoothly. But in Abaqus, beam radii are associated with beams and it means each beam has its own constant radius. The conversion between these two brought errors. To evaluate this conversion accuracy, the same design process indicated in the Figure 1 was implemented but without optimization. It was: “tapered beams model in Element Free → Constant beams in Abaqus (with no optimization)→ Tapered beams in Element Free”. In this workflow, a bike stem model with over 10000 tapered beams were tested.

The conversion errors of each nodal radius were plotted in Figure 5. It calculated the percentage of the beam thickness difference between the final Element Free model and the original Element Free file. The mean value of all absolute error is 0.46%, which is acceptable. A few large conversion errors were found at nodes where small number (i.e. two) of beams were connected. This error can be improved in each specific case by choosing different interpolation algorithms and rules.

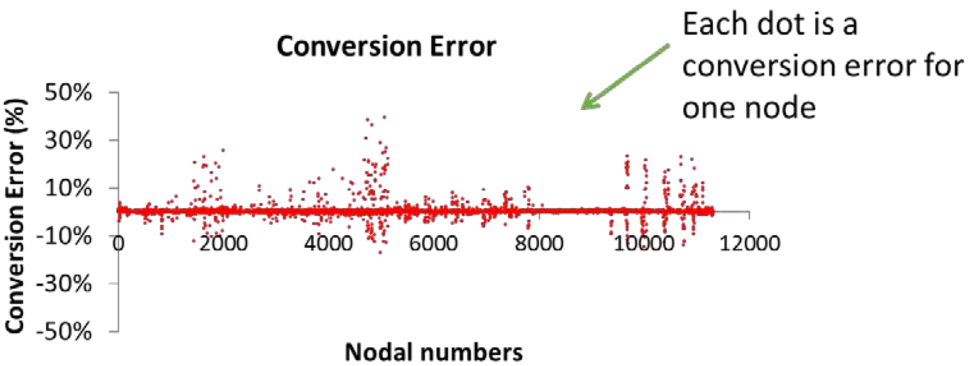

Figure 5. Conversion Accuracy of beam radii at nodes.

4. Discussion

With the combination of structure optimization tool SIMULIA Tosca and the lattice design software Element Free, we are able to design and optimize complex lattice structures and create STL files for additive manufacturing. This workflow produced lattice structures with large mechanical stiffness within light weight constraint. The conversion accuracy test showed that the model transfer between these two software brought only minor errors. Moreover, the strong meshing function of Element Free provided a way of what you see from Tosca is what you get as an STL file.

As an extension of this work, in the case of many repeated cell lattice structures are present, Tosca Structure is able to do a Representative Elementary Volume (RVE) design. Starting from a bulk of material with periodic boundary conditions applied, Tosca can produce a light-weight but stiff RVE lattice structure based on different loading conditions.

5. References

1. Bendsoe, M. P., Martin P., and Sigmund, O., "Topology optimization: theory, methods and applications", Springer, 2003.

2. Gerzen, N, Peter M. C., and Claus B. W. P., "Sizing optimization for industrial applications and best practice design process". High Performance and Optimum Design of Structures and Materials II 166, 41, 2016.

3. Vasiliev, V.,V., Vyacheslav A. B., and Alexander F. R. "Anisogrid composite lattice structures–Development and aerospace applications." Composite structures 94.3: 1117-1127, 2012.

4. ISO-4210-5, "Cycles - Safety requirements for bicycles - Part 5: Steering test methods". 2014.

5. Abaqus Users Manual, Version 6.10-1, Dassault Systémes Simulia Corp., Providence, RI.

6. Element Free Manual, http://www.ntopology.com/glossary/, nTopology .inc, 2017.

7. TOSCA structure Manual, Version 2016, Dassault Systémes Simulia Corp., Providence, RI,