Finite Elements in Analysis and Design 138 (2018) 1–11

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

journal homepage: www.elsevier.com/locate/finel

Modeling and simulation of large, conformal, porosity-graded and lightweight lattice structures made by additive manufacturing

Patrick Terriault [*], Vladimir Brailovski

Mechanical Engineering Department, École de technologie supérieure, 1100 Notre-Dame Street West, Montreal, Quebec, H3C 1K3, Canada

ARTICLE INFO

Keywords:
Lattice structure
Finite elements
Porosity graded
Conformal topology
Additive manufacturing

ABSTRACT

The emergence of additive manufacturing has enabled the design and manufacture of lightweight parts such as lattice structures with a complex geometry. However, conducting computer simulations to predict the mechanical behavior of such structures remains a challenge, especially when the number of struts is high. Firstly, this paper presents an approach to quickly build a lattice structure made of a large number of struts. The method takes advantage of the capabilities of finite element meshers to generate the tessellation of the volume into a series of tetrahedrons whose edges define the struts of the lattice. A graded-porosity can be defined during this process. Secondly, for the numerical simulation, the struts are modeled by beam finite elements. To adequately simulate the entire range of porosity, the material stiffness can be adjusted at any point of the structure, mainly to compensate for the lack of validity of stout beam elements. A numerical validation demonstrates the validity of the method by adequately predicting the displacement distribution of a mechanically loaded lattice structure, but an experimental validation raises issues related to the manufacturing limitations of small struts.

1. Introduction

The additive manufacturing of metallic parts opened the door to the design and optimization of lightweight structures [1], and the use of lattice cells produces high-strength structures accompanied by a relatively low mass [2]. Many different types of cells can be used in lattice structures, such as tetrakaidecahedral [3–5], cubic [6,7] and diamond [8,9]. Predictions of the mechanical behavior of such lattice structures can rely on analytical methods or on numerical ones such as finite elements. In an analytical model, Bernouilli-Euler or Timoshenko beam theories can be used to predict the behavior of a single cell; indeed, several types of unit cells (cubic, tetrakaidecahedral, octahedral and diamond) were studied using these theories [6,10–13]. The main limitation of this technique is the validity of the beam theories, as they force the length of the strut to be significantly greater than its depth.

As a consequence, only very high porosity structures can be studied analytically. A single cell can also be studied with solid finite elements, in which case the span/depth ratio limitation no longer holds. Several such works have been carried out for tetrakaidecahedral, cubic and spherical unit cells [7,14,15]. In these cases, the entire range of porosities can be studied, but the number of solid elements quickly becomes an issue. Indeed, as discussed in this paper, the number of nodes and elements required to properly model a unit cell with solid elements is so large that an entire lattice structure with many struts is simply unmanageable in terms of computing time and memory. Relatively few works have studied the behavior of an entire structure comprising a significant number of struts; these include a simulation of the penetration of an indenter into a tetrakaidecahedral lattice structure [5] and an analysis of the effect of a notch on a similar structure [16]. In such situations, beam elements representing high porosity structures can be simulated, and moderate to low porosity ones set aside.

The primary objective of this paper is to present a method that can use beam finite elements to simulate an entire lattice structure with no porosity level limitation. Its main feature is that it corrects the material stiffness to compensate for the loss of validity of the beam elements theory when the span/depth ratio of the element becomes too small. The lattice structures studied in this work are 3D-trusses composed of a series of struts connected together to define tetrahedral cells. These were selected because such structures reveal their superior mechanical performances in light-weight structures [17].

The term “large” in the title does not refer to the physical size of the object being simulated, but rather to the number of struts in the lattice structures. Indeed, the method was developed to simulate, using a conventional desktop computer, structures with tens or hundreds of thousands of struts. The title also includes the term “conformal” because the suggested method guarantees that the extremities of the cells will match the physical boundaries of the part.

* Corresponding author.
E-mail address: patrick.terriault@etsmtl.ca (P. Terriault).

https://doi.org/10.1016/j.finel.2017.09.005
Received 2 December 2016; Received in revised form 2 August 2017; Accepted 21 September 2017

The first section of this paper gives a brief overview of the entire developed method. The second section describes how the STL file of the porosity-graded truss structure is generated for an additive manufacturing process. The ability to efficiently produce the STL file of a complex 3D lattice structure is indeed the secondary objective of this work. Convergence analyses are then undertaken in the third section to determine the optimal finite element size that produces converged results for both the solid and beam elements used in this work. In the fourth section, a stiffness correction factor is introduced to match the results of a finite element simulation using beam elements with those using solid elements, even for low porosities. The use of the stiffness correction factor is numerically validated in the fifth section by comparing the mechanical behavior of a lattice structure modeled by either beam or solid elements. Finally, in the sixth section, a structure made of a large number of struts is modeled, built by additive manufacturing, simulated by finite elements, and experimentally tested in order to verify the validity of the method involving a complex structure.

2. Overview of the method

This work is based on the tessellation of a part into a series of tetrahedrons from which the 3D truss structure is defined. Such a tessellation process has now been mastered by a large number of finite element meshers available on the market; the ANSYS 15.0 mesher is used here. With their preferred the CAD software, the users must first build the geometry of the part that defines the volume to be filled with the lattice structure (see Fig. 1A). The geometry is then transferred to the mesher via an exchange file format such as IGES or STEP, and the tessellation is carried out by controlling the size of the tetrahedrons (see Fig. 1B). The result is inevitably a conformal topology. Each tetrahedron is made of 4 vertices and 6 edges that can be shared with other tetrahedrons. Each edge defines a strut of the lattice structure. In some particular circumstances, the triangular “holes” that are generated on the external faces of the part could be sealed, for example in the case where some faces of the truss structure must withstand a pressure. Such “closed” faces selected by the user are colored in red in Fig. 1B. All the other faces are opened specifically for the removal of the powder trapped inside the part during the manufacturing process.

The other important feature is the possibility of adjusting the porosity at every vertex of the structure. As seen by the size of the openings in Fig. 1C, an increasing porosity from the left to right side of the part has been defined. The local porosity is specified by the user at a few selected vertices only. The porosity at all other vertices is interpolated by kriging, an interpolation technique that has been widely used in geology for decades [18].

Coordinates of the vertices and tessellation connectivity are then exported into text files and read by a proprietary MATLAB program that reconstructs the structure and generates two outputs. The first output is the STL geometry, which can be used either for solid finite element analyses or for additive manufacturing (see Fig. 1C). More details concerning the STL representation of the part are given in the next section. The second output is a finite element model for which the struts are modeled by beam elements and the closed faces by shell elements (see Fig. 1D). The finite element model is used to simulate a structure comprised of a large number of struts.

3. Modeling of tetrahedral cells

This section describes the approach used to model the tetrahedral cells for the STL file. The process follows the tessellation step, after which the coordinates of the vertices and the connectivity of the edges are known for each tetrahedron (see points i, j, k and l of Fig. 2A for an illustration of the vertices of a tetrahedron). Also, at each vertex, the

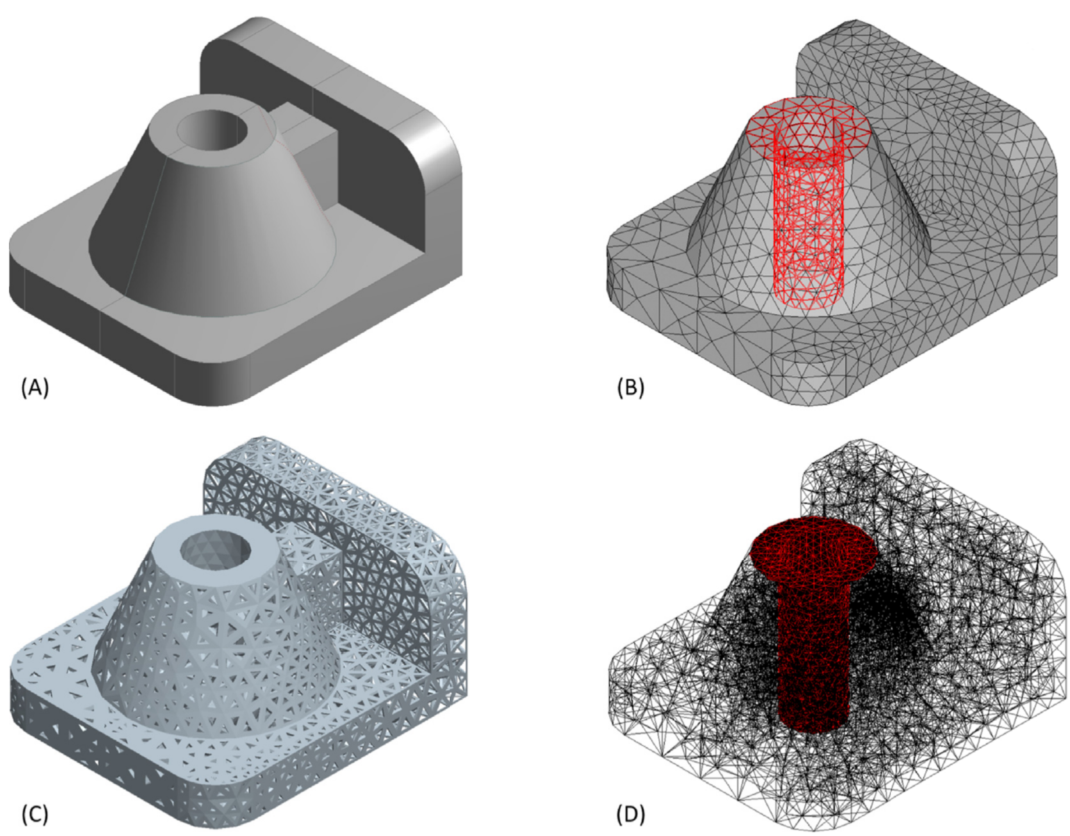

Fig. 1. A) Envelope of the part to be filled by a lattice structure, B) Tessellation of the volume with tetrahedrons and identification of the closed faces represented in red, C) Geometry represented in the STL file with an increasing porosity from left to right, and D) Finite element model made of beam and shell elements. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)