**PAPER** • **OPEN ACCESS**

# Finite element analysis of closed-cell aluminum foam approximated with Weaire-Phelan unit cell structure

To cite this article: M File *et al* 2022 *IOP Conf. Ser.: Mater. Sci. Eng.* **1246** 012002

# Finite element analysis of closed-cell aluminum foam approximated with Weaire-Phelan unit cell structure

**M File[1,3], R Cseke[1], D Huri[1,2], G Balogh[1] and T Mankovits[1]**

[1] Department of Mechanical Engineering, Faculty of Engineering, University of Debrecen, H-4028 Debrecen, Ótemető u. 2-4., Hungary
[2] Doctoral School of Informatics, Faculty of Informatics, University of Debrecen, H-4028 Debrecen, Kassai u. 26., Hungary
[3] Author to whom any correspondence should be addressed.

**Abstract.** The finite element modelling of metal foam structures have proven to be a difficult task. Many different modelling approaches exist, both for creating the CAD model and for performing the finite element simulations. This article details the CAD and finite element modelling of an existing aluminum foam. To model the cell structure, the Weaire-Phelan unit cell was used. The different parameters of the unit cell were adjusted to fit the parameters of the existing foam. The physical properties of the finished model were then compared to the original specimen to validate its accuracy. After the comparison, it was determined, that the Weaire-Phelan unit cell can be used to model the physical properties of the structure. The validated and simplified model was then used to perform a finite element analysis under compressive loading. The results were compared to the compression test results of the existing aluminum foam. By comparing the results and calculating the relative errors, it was determined that both the physical and the mechanical properties of metal foam structures can be modelled using this exact method with significant accuracy.

## 1. Introduction

Closed-cell metal foams are used in a wide range of structural elements, usually the most common load case is compression. The reason of the usage is the lightness, the high compression strength and the high energy consumption under compression [1]. During the design phase, modelling the behaviour of these materials is a very important task. The behaviour of these materials under compressive loading is detailed in [2–4].

Many different geometrical modelling approaches exist, one of which is the μCT based two-or three-dimensional model generation. Two dimensional models were constructed using this method in [5,6]. Closed cell foam materials were modelled using three-dimensional models generated based on μCT images and analysed using finite element software in [7,8]. This method enables the creation of the most accurate model. However, it requires special equipment and is a time-consuming process. Another method is the usage of uniform cells to simplify the internal geometry. A combination of spherical and cruciform-shaped cells was used to model closed-cell aluminum foam in [9]. Different, already existing unit cells, such as Kelvin and Weaire-Phelan unit cells can also be used to model closed-cell metal foam materials, as presented in [10,11]. A comparison of these unit cells can be seen in [12].

This paper deals with the be investigation of the accuracy of the Weaire-Phelan unit cell, when modelling a closed-cell aluminum foam specimen for compressive loading. A specimen was modelled

using these unit cells. The accuracy of the created CAD model was validated by comparing its physical properties to the original specimen. The validated model was used for the finite element simulations. By comparing the finite element simulation results to the laboratory measurement, the accuracy, and the usability of the simplified model were be determined. During the research only the elastic region was tested. The model could be expanded to the modelling of the plastic deformation zone as well in the future.

## 2. Materials and methods

### *2.1. Parameters of the modelled metal foam*

The closed-cell aluminum foam was procured from Aluinvent Zrt. For the compression test five cylindrical specimens were cut according to the ISO 13314 standard [13]. The diameter and the length were 30 mm, respectively. Table 1 shows the average physical properties of the material. The porosity was calculated based on the weight of the specimens. An image about the specimen is shown in figure 1. The average cell size and wall thickness was determined using CT analysis.

**Table 1.** The properties of the aluminum foam

| Property | Value |
|---|---|
| Weight (average) | 6.6 g |
| Porosity (average) | 89.27 % |
| Volume (average) | 2389 $mm^3$ |
| Young's modulus | 2374 MPa |
| Poisson's ratio | 0.29 |

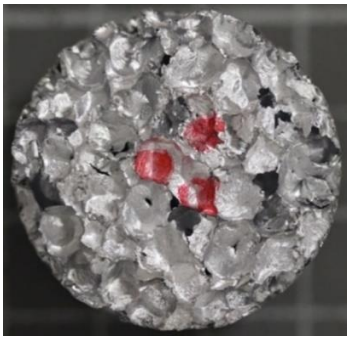


**Figure 1.** One of the specimens used for the modelling

### *2.2. Modelling the specimen using Weaire-Phelan unit cells*

The 3D model was created using the SIEMENS Solid Edge software. For the cell structure approximation, the Weaire-Phelan unit cell was chosen [12]. After several steps the unit cell was modelled, see figure 2.

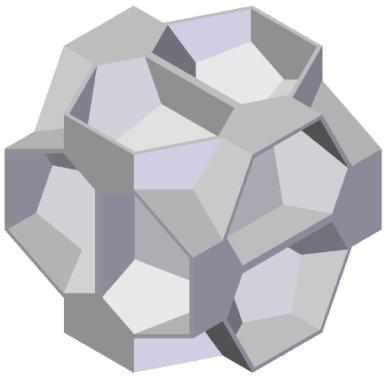
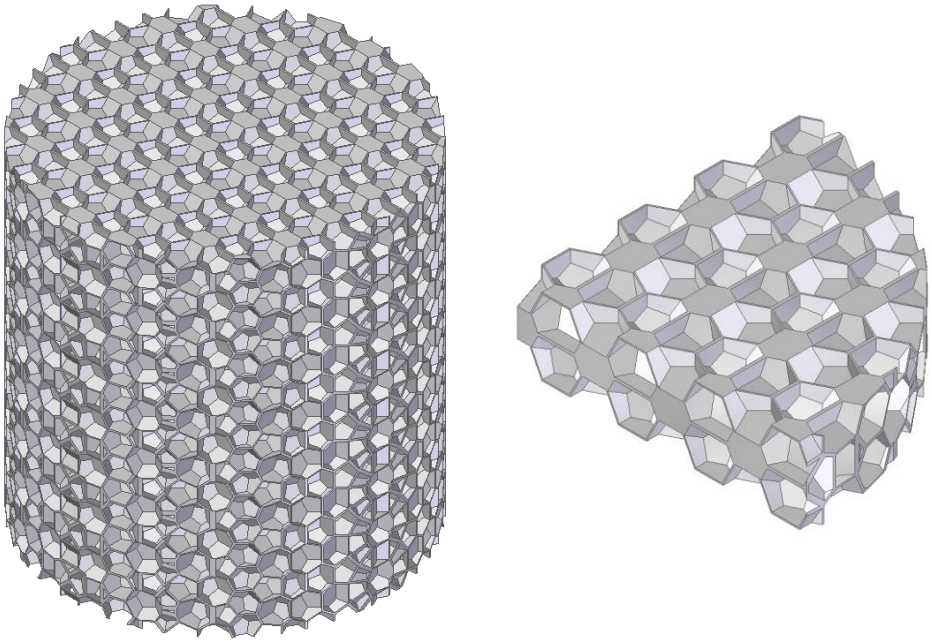


**Figure 2.** Weaire-Phelan unit cell modelled using CAD software

The unit cells were multiplied to create a cube with 30 mm edge length which was then cut to the cylinder shape of the specimen with a 30 mm diameter. The final CAD model of the aluminum foam specimen can be seen in figure 3 (a). Finally, this model was cut into a quarter model with one layer of unit cells, to prepare it for the finite element simulations, which can be seen in figure 3 (b).



(a) (b)

**Figure 3.** (a) CAD model of the aluminum foam specimen; (b) one-layer high quarter model prepared for the finite element simulations

*2.3. Finite element analysis*

The finite element simulations were done using the Simcenter Femap with its integrated NX Nastran solver. After opening the prepared 3D model with the software, the material model was created. Since in this work only the elastic region of the material was analysed, the default linear elastic material model was used. The data required to create this model was the Poisson's ratio and the Young's modulus of the material, seen in table 1. A prescribed displacement of 0.0074 mm (0.2% strain) was applied to the top surface of the specimen as loading. Since a quarter model was used, the symmetry planes were constrained in their normal direction. The last remaining axis, the Y axis was constrained on the bottom surface and on the surface where the load was applied as well, as the software requires it for prescribed displacement.

A mesh convergence analysis was then performed to eliminate the mesh dependence of the simulated results. The force values were used as the mesh convergence criteria, relative error values were calculated between the maximum force values and convergence was detected if three deviation values were smaller than 0.5%. The maximum force values of the mesh convergence analysis are plotted in figure 4 and the calculated deviation values can be seen in table 2.

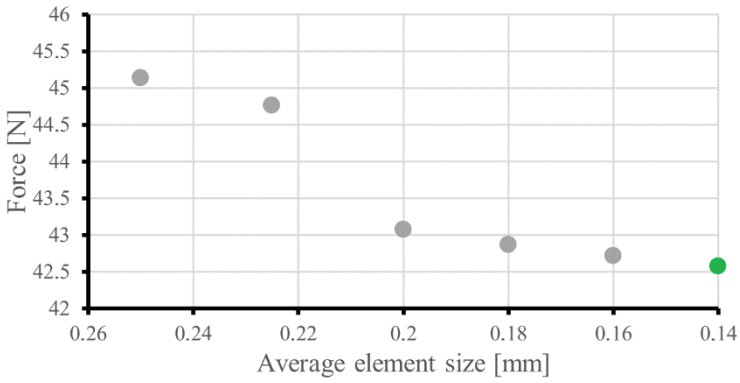



**Figure 4.** Results of the mesh convergence analysis

**Table 2.** Relative deviation values of the mesh convergence analysis

| Element size [mm] | Force [N] | Relative deviation |
|---|---|---|
| 0.25 | 45.132 | - |
| 0.225 | 44.758 | 0.82868 % |
| 0.2 | 43.07159 | 4.5653 % |
| 0.18 | 42.8624 | 0.48568 % |
| 0.16 | 42.71221 | 0.3504 % |
| **0.14** | **42.567** | **0.33997 %** |

The analysis has conducted that the 0.14 mm average element size was optimal for the tetrahedral solid elements. The parameters for the final discretisation can be seen in table 3 and the meshed finite element model can be seen in figure 5.

**Table 3.** The parameters of the selected mesh

| Property | Value |
|---|---|
| Element type | Solid |
| Element shape | Tetrahedral |
| Nodes per element | 4 |
| Element size | 0.14 mm |
| Number of elements | 334252 |

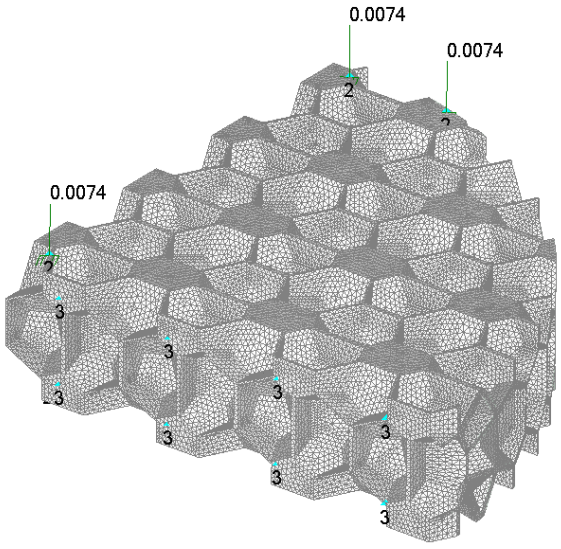



**Figure 5.** The meshed finite element model

*2.4. Laboratory measurement*

The laboratory measurement was done using the INSTRON 68TM-10 universal material testing system and the strains were measured using the INSTRON AVE2 advanced video extensometer. Similarly to the finite element simulations, the specimens were also compressed until 0.2% strain which in the case of the whole specimen is 0.06 mm. The load was applied with a 1 mm/min velocity. The results of the 5 compression tests were then summarised and the median value was selected for the comparison with the results of the finite element analysis.

## 3. Results and discussion

*3.1. Comparison of the CAD model and the specimen*

To determine the accuracy of the CAD model, its different properties were compared to the properties of the original foam specimen. Table 4 shows the comparison of the physical properties.

**Table 4.** Comparison of the aluminium foam specimen and the CAD model based on the physical properties

| Property | Aluminum foam specimen (average values) | CAD model | Difference |
|---|---|---|---|
| Weight | 6.6 g | 6.901 g | 4.560 % |
| Porosity | 89.27 % | 86.43 % | 3.181 % |
| Volume | 2389 $mm^3$ | 2421 $mm^3$ | 1.339 % |

It is visible from the less than 5 % difference values, that the model is an accurately approximating the physical properties of the original specimen.

*3.2. Results of the finite element analysis*

During the evaluation of the finite element results, the force values were in focus. The simulation has conducted that compressing the specimen by 0.2% (0.0074 mm) takes 42.567 N of force. Since a one-layer quarter model was used, this force value must be multiplied by 4, meaning the force for the full specimen was 170.268 N, and the displacement must be scaled up to reflect the 0.06 mm compression of the whole specimen.

*3.3. Results of the compression tests*

The compression tests were also evaluated until the 0.2% strain value, which in the case of the whole specimen was 0.06 mm. From the 5 tested specimen, the median value was selected. The maximum load on this specimen was 162.608 N, the results of the 4 other specimens showed a 3.56% maximum deviation from the median value.

*3.4. Comparison of the finite element analysis and the laboratory measurement*

The results of the finite element simulation and the laboratory measurements were compared. The relative errors were determined, thus the accuracy of the created model was evaluated. The comparable factor was the maximum force value at the 0.06 mm (0.2%) compression. For the relative error calculation, the following equation was used:

$$e_r = \frac{v_{simulated} - v_{measured}}{v_{measured}} \cdot 100 \quad (1)$$

where $e_r$ is the relative error, $v_{simulated}$ is the maximum force evaluated from the simulation, and $v_{measured}$ is the maximum force evaluated from the measurement. The evaluated results with the calculated relative errors can be seen in table 5.

**Table 5.** The calculated error value between the results of the finite element simulation and the laboratory measurement

| Value | Finite element simulation | Laboratory measurement | Relative error |
|---|---|---|---|
| Maximum force | 170.268 N | 162.608 N | 4.7 % |

From the measurement and the simulation, a force-displacement curve was also evaluated, which can be seen in figure 6.

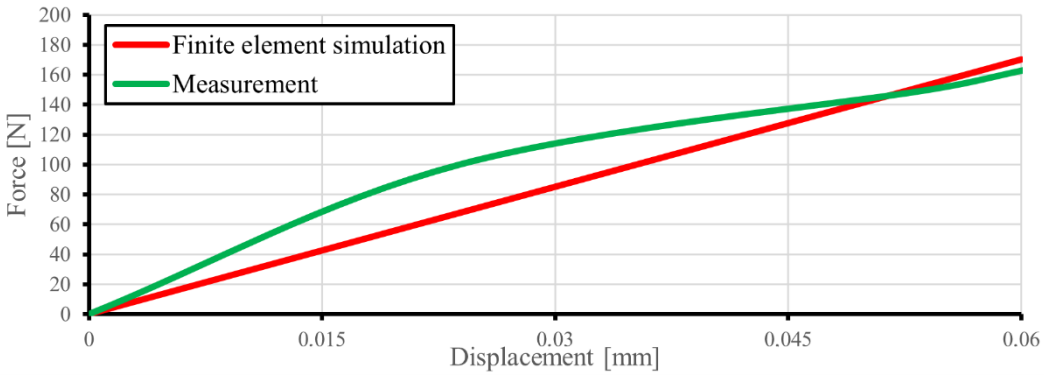



**Figure 6.** The force-displacement curves of the measurement and the simulation

## 4. Conclusion

This article introduced the modelling and finite element simulation of a closed cell aluminum foam specimen under compressive loading. The structure of the material was approximated by Weaire Phelan unit cells, which were then multiplied and shaped to match the starting foam specimen. The created CAD model was then simplified into a one-layer quarter model, to prepare it for the finite element simulations. Before the simulations, the CAD model was validated by comparing its physical properties to the original specimen. The comparison has shown that the Weaire-Phelan unit cell structure is able to model the physical properties of the material with significant accuracy, as the difference values have been less than 5 % in all cases. After the validation, the finite element simulations were performed on the simplified model. The boundary conditions and the load were given in such a way, to recreate the laboratory conditions as closely as possible. The simulated result was compared to the measured data and relative error values were calculated, to find out the accuracy of the created model. It is visible from the less than 5% relative error values, that the Weaire-Phelan unit cells and the applied finite element analysis can simulate the elastic behaviour of closed cell aluminum foam materials accurately. The use of the Weaire-Phelan unit cell can shorten the design process of structures using metal foam materials, as the creation of the structure only requires the multiplication of one unit cell, while being able to model the physical properties and the mechanical behaviour of these materials with sufficient accuracy.

## Acknowledgments

The research was supported by the Thematic Excellence Programme (TKP2020-NKA-04) of the Ministry for Innovation and Technology in Hungary, within the framework of the (Automotive Industry) thematic program of the University of Debrecen.
We thank Aluinvent Zrt. for providing the closed-cell aluminum foam for the research.

## References

[1] Ashby, M, Evans, A, Fleck, N, Gibson, L, Hutchinson, J, Wadley, H, et al. Metal Foams: A Design Guide. Appl Mech Rev. 2001 Nov 1;54(6):B105–6.
[2] Hu Y, Fang Q-Z, Qian J. Effect of cell structure on the uniaxial compression properties of closed-cell foam materials. Mater Today Commun. 2021 Mar;26(February):102104.
[3] Szlancsik A, Norbert Orbulov I. Compressive properties of metal matrix syntactic foams in uni- and triaxial compression. Mater Sci Eng A. 2021 Oct;827:142081.
[4] Szlancsik A, Katona B, Dombóvári Z, Orbulov IN. On the effective Young's modulus of metal matrix syntactic foams. Mater Sci Technol. 2017 Dec 12;33(18):2283–9.
[5] Caty O, Maire E, Youssef S, Bouchet R. Modeling the properties of closed-cell cellular materials from tomography images using finite shell elements. Acta Mater. 2008

Nov;56(19):5524–34.
[6] Kádár C, Maire E, Borbély A, Peix G, Lendvai J, Rajkovits Z. X-ray tomography and finite element simulation of the indentation behavior of metal foams. Mater Sci Eng A. 2004 Dec;387–389(1-2 SPEC. ISS.):321–5.
[7] Saadatfar M, Mukherjee M, Madadi M, Schröder-Turk GE, Garcia-Moreno F, Schaller FM, et al. Structure and deformation correlation of closed-cell aluminium foam subject to uniaxial compression. Acta Mater. 2012 May;60(8):3604–15.
[8] Hangai Y, Yamaguchi R, Takahashi S, Utsunomiya T, Kuwazuru O, Yoshikawa N. Deformation behavior estimation of aluminum foam by X-ray CT image-based finite element analysis. Metall Mater Trans A Phys Metall Mater Sci. 2013;44(4):1880–6.
[9] Hasan MA. An Improved Model for FE Modeling and Simulation of Closed Cell Al-Alloy Foams. Adv Mater Sci Eng. 2010;2010:1–12.
[10] Chen Y, Das R, Battley M, Xu Z. Compressive and shear strengths of the ductile closed-cell Kelvin and Weaire-Phelan foams along the lattice direction [100]. Thin-Walled Struct. 2018;132(November 2017):237–49.
[11] Vengatachalam B, Poh LH, Liu ZS, Qin QH, Swaddiwudhipong S. Three dimensional modelling of closed-cell aluminium foams with predictive macroscopic behaviour. Mech Mater. 2019 Sep;136(August 2018):103067.
[12] Latré SK, De Pooter S, Buffel B, Brabazon D, Seveno D, Desplentere F. Comparative study of a cubic, Kelvin and Weaire-Phelan unit cell for the prediction of the thermal conductivity of low density silica aerogels. Microporous Mesoporous Mater. 2020 Jul;301(March):110206.
[13] ISO. Mechanical testing of metals - Ductility testing - Compression test for porous and cellular metals. Int Stand. 2011 [cited 2022 Jan 24];ISO 13314(1).