Open Cell Metal Foams: Measurement and Numerical Modelling of Fluid Flow and Heat Transfer

J. Ettrich¹,², A. August¹,², B. Nestler¹,²,
¹Karlsruhe University of Applied Sciences, Institute of Materials and Processes, Moltkestrasse 30, 76133 Karlsruhe, Germany
²Karlsruhe Institute of Technology, Institute of Applied Materials, Haid-und-Neu-Str. 7, 76131 Karlsruhe, Germany

Abstract

We herein present results from experimental investigations and new numerical methods for the investigation of fluid flow and heat transfer in cellular solids, in particular open cell metal foams. Results from a custom made test facility contribute to enlarge the public available database of open cell metal foams and serve for the validation of applied commercial and in-house numerical methods. Contrary to the classical numerical finite volume (FV), finite difference (FD) or finite element (FE) interface tracking methods, the employed method combines a lattice Boltzmann solver for fluid flow with a heat transfer equation, both in context of a diffuse interface representation of the complex cellular structure [1]. The diffuse interface concept (also known as phase-field modelling) is used to smoothly generate and map complex microstructures [2]. We assume no interface dynamics, i.e. the order parameter of the phase field is only used to distinguish between different phases, namely fluid and solid. However, in view of multiphysics applications and more complex convoluted microstructures, extensions to phase change systems can hardly be mastered by body fitted, interface tracking methods. The presented method promises to be a comfortable, feasible and high potential method.

Keywords: Phase-Field, Lattice-Boltzmann, Heat-Transfer, Cellular Solids, Open Cell Metal Foam

1 Introduction

There has been a growing interest in cellular solids in engineering applications, together with energy efficiency and thermal balance of machinery, which are the issues of today. Metal foam, which belongs to the class of porous media, was discovered in engineering applications during the last decades. The physical, mechanical and thermal properties of open-cell metal foams are similar to the properties of pure solids. However, their uniqueness is different in terms of range and combination, which enables the materials to be utilised in numerous engineering applications and innovative products.

An analysis recently derived for heat transfer applications of matrix and foam materials, is available in [3]. In [4], the achievement of finned heat exchangers and open-cell metal foam heat exchangers can be found. In [5 - 9], experimental measurements are carried out on a vast
number of aluminium and copper foams, regarding pressure loss and transfer of heat. The characteristics of nickel in terms of pressure loss, however, are examined in [10]. The references [11] and [12] deal with ceramic foam structures, with regard to the utilisation in catalytic and chemical applications.

![Figure 1](image_url)

Figure 1. General view on the final assembly of the custom made test rig for fluid flow and heat transfer measurements of open cell metal foams [24].

A detailed description of the numerical treatment of the fluid flow and heat transfer in cellular solids on a high resolution pore scale level is presented in e.g. [13-18]. The encouragement and inspiration for the development of a simple, smooth and coherent method is also motivated in these publications.

This work has two intentions. On the one hand, the results of the experiments are added to the accessible database of open-cell metal foams referring to their hydraulic and thermal properties, and on the other hand, the modelling attempts are devoted to the creation and improvement of suitable numerical approaches by using the phase field method of the diffuse interface as their background.

In order to solve the fluid flow in this work, a lattice Boltzmann type method is employed, which has become a powerful numerical tool comparable to classical methods based on computational fluid dynamics. It can be used for computational studies of incompressible and quasi-incompressible flow regimes. However, the number of publications on Lattice Boltzmann models for fluid flow, combined with the phase field method as background, has been very rare throughout the last decade. While the publications [19-21] deal with the simulation of dendritic solidification and the effects of liquid motion, the papers [22, 23] focus on the simulation of fluid flow in complex microstructures, which is dissimilar to the classical field of application of a phase field method. The evolution equation of temperature is solved within the scope of the phase field method, with regard to heat transfer simulations. During this work, a new, segmented tensorial formulation is developed because of the numerical treatment, which enables three-dimensional, transient temperature fields to be solved in a stable and accurate way.
2 Experiments

In [23, 24] present an entirely new test rig, that is designed with regards to accurate heat transfer and fluid flow measurements of open cell metal foam samples, cf. fig. 1 Experimental studies are carried out on circular 10 ppi, 20 ppi and 30 ppi open cell foam samples made from aluminium and copper up to turbulent flow regime. Specifically designed custom made adjustable pressure sensors and temperature rakes are utilized, each equipped with high accuracy sensors, cf. fig. 2.

The overall length of a cylindrical encapsulated sample is limited due to the demolding. According to the scheduled porosities, the maximum sample length is limited to 20 mm, because the foam samples as well as the isolated test section is designed modularly, and permits a maximum sample length of 100 mm. Presuming that the heat conduction in axial direction is negligible compared to the radial conduction and convective effects, we expect that the segmentation will still provide valuable results.

![Figure 2](image)

*Figure 2.* Details of the test rig: (a) temperature rake (b) view on the opened test section with foam samples and heat sources, (c) cross sectional view on the design data of the measurement devices and the test section.

A fully automated governance and control system is realized using the system design platform LabView. It permits the simultaneous recording of all transient signals as well as the automatic steering of the adjustable pitot tubes and the fully automatic control of the heat sources. The latter enables to keep temperature at cylindrical shroud of the samples at a constant value in flow direction and at ±1 °C within a temperature range of about 30 °C up to 120 °C at full discharge.

As exemplary results of the experimental investigation, figs. 3 depict the measurement data and the established pressure loss and Nusselt number correlations. In accordance with a study on ceramic foams [11], the pressure loss correlation is formulated in terms of the nondimensional quantities Hagen number Hg and the Reynolds number Re, whereby the correlation for ceramic foams is also given in 3(a) for comparison. For the Nusselt correlation (solid lines in 3(b)) the common ansatz \( \text{Nu} = C \cdot \text{Re}^{m} \cdot \text{Pr}^{1/3} \) is chosen, and the coefficient \( C \) and exponent \( m \) are correlated to pore scale measures [24].
3 Numerical Modelling

3.1 Diffuse Interface

Phase field models evolved from the challenge to solve free boundary problems mathematically, employing direct numerical methods. In comparison to the classical front tracking methods, phase boundaries are not explicitly treated, boundary conditions at the interface between different medias are naturally included in the solution of the respective partial differential equation. Thus, the phase field approach can be classified as an interface capturing method, but furthermore it permits multidisciplinary couplings to various physical models. A unique characteristic of the phase field method is that it permits the seamless numerical simulation of interface motion and complex geometrical evolution and, moreover, problems with catastrophic phase terminations.

In the course of this work, we employ the phase field method based on the formulation of [1]. For the simulation of fluid flow and heat transfer in open cell metal foams, we limit the consideration to a two-phase system, namely the solid and the fluid phase. The phase field specific order parameter is $\phi_{\text{solid}} = 1$ in the bulk solid and $\phi_{\text{fluid}} = 1$ in the bulk fluid phase, whereas it becomes 0 in the opposite phase. At the interfaces of finite width between the phases, the order parameter changes smoothly between 1 and 0. For simplicity, no phase dynamic such as an evolution of the phase states is present. Consequently, we do not solve for the phase field equations, but the geometry of the cellular solid is specified by the spatial distribution of the order parameter, which partitions the overall computational domain into areas of pure solid and pure fluid.

For the digital representation of the complex structure of the cellular solid we utilize the algorithm of [2], which is presented in more detail in a second contribution [25].

3.2 Fluid Flow

According to the kinetic theory of gases [26, 27], the lattice-Boltzmann method (LBM) is a numerical scheme to solve the discretised Boltzmann transport equation. Thereby, the governing equations of fluid mechanics can be recovered from the LBM by applying the Chapman-Enskog expansion [28]. The method is chosen because, compared to other fluid dynamics
simulation approaches, it captivates by its simplicity in terms of modeling the physical process and handling of complex geometries as well as implementation. Additionally, the LBM is highly appropriate for parallelisation, which is an important property aiming towards large scale and massive parallelisation [29]. In addition to the popular single relaxation time (SRT) model [30], the multiple relaxation time (MRT) approach as proposed in [31] is implemented in the phase-field simulation package PACE3D. Furthermore, to allow for the simulation of high Reynolds number turbulent flows, we employ the subgrid-scale model of Smagorinsky [32–34]. While focusing on the comparison of experimental and simulation results in the subsequent sections, more details and validations are presented in [24, 25].

3.3 Heat Transfer

According to the work of [35] we use an extended tensorial mobility formulation of the temperature evolution equation in order to accurately solve the heat transfer within the context of a diffuse interface realisation. A detailed description together with the results of a number of one-, two- and three-dimensional validations are given in [36, 37], which show excellent agreement with analytical and numerical reference solutions.

![Figure 4](image-url)

**Figure 4.** (a) Computational domain, representing a quarter of a cylindrical open cell metal foam sample of diameter 40 mm and length 20 mm. (b) Distribution of the order parameter on a cross section of the numerical domain, showing regions of pure solid (red), pure liquid (blue) and the highlighted interfacial regions.

4 Comparison of experiment and simulation

In this section we present the results of the coupled lattice Boltzmann and segmented tensorial heat transfer solvers outlined above, in comparison to experimental data. In order to save computational resources and to speed up the simulations, the numerical model represents a quarter of a cylindrical open cell metal foam sample of diameter 40 mm and length 20 mm, cf. fig. 4. The porosity is processed as control measure to generate a realistic structure based on the pore scale measures taken from the foam characterisation.

The integral pressure loss for a foam sample with a pore density of 10 ppi and a porosity of 88.44% for different Reynolds numbers is given in fig. 5(a) in comparison with experimental
data and a numerical reference solution obtained from StarCCM. The numerical results obtained with the diffuse phase boundary representation in PACE3D are all within an error band of about ±0.05% for a full scale of 25hPa, which corresponds to the accuracy of the pressure transducer, indicated by the dashed lines. The results of StarCCM show slightly higher deviations for increasing Reynolds number, whereas the results obtained with PACE3D are in good agreement with the measurement data.

![Figure 5: Comparison of simulation results and experimental data for (a) pressure loss and (b) heat transfer for a 10 ppi aluminium foam sample [24].](image)

In addition, a series of coupled fluid flow and heat transfer simulations is conducted to compare the integral thermal performance by means of the Nusselt number for different Reynolds numbers, cf. 5(b). The results are all within an error band ±5% of the full scale value. The overall agreement of the numerical and experimental reference values is excellent for the contemplated range of Reynolds numbers.

Besides the comparison of integral performance data, the numerical studies provide a detailed insight, which can hardly be accessed by experimental methods. As an example, fig. 6(a) shows streamlines in front of a foam edge connected to the circular shroud, where a horseshoe vortex establishes and influences the local heat transfer. The overall impact of the flow field becomes immediately obvious in fig. 6(b), where the temperature distribution is given along the entire surface of the foam structure.

### 5 Outlook and conclusion

The experimental measurements on open cell metal foams facilitate pressure loss and heat transfer characteristics up to a high Reynolds number, turbulent flow regime.

The data contribute to the existing data basis for cellular solids in the open literature, and moreover serve as validation for a new numerical simulation approach. Dedicated methods for the simulation of fluid flow and heat transfer are implemented in a modern phase field approach. Compared to classical methods, the diffuse surface profile enables to computationally resolve convoluted microstructures and, allows for further investigations on more complex multiphysics applications including phase evolution and phase transformations.

The application of the new simulation method to algorithmically generated foam structures, reflecting various material properties on a micrometer length scale, shows excellent agreement of integral performance data. The simulations provide valuable insights into local flow...
and heat transfer effects within the microstructure which are usually not available when conducting experiments. Thus, this approach shows high potential when it comes to the issues of structure optimization and material design.

6 Acknowledgements

The authors thank the scientific staff at the Institute of Materials and Processes at the Karlsruhe University of Applied Sciences for their support and work on the PACE3D package. Support provided by the Federal Ministry of Education and Research (BMBF) through the FHPProfUnt founding project SimFoam (FKZ 17029X10) is also greatly acknowledged. We are grateful for financial support through the Helmholtz POF III (program oriented funding) within the program EMR (energy efficiency, resources and materials).

7 References


[27] Haenel, D., Molekulare Gasdynamik. Berlin, Heidelberg : Springer Verlag, 2004


