Dear Prof. Lynn Walker,

we would like to thank you and the two referees for evaluating our manuscript. We have addressed the reviewers' concerns and believe that the quality of the manuscript has improved considerably. If need be, we would be happy to make any further corrections.

To ease the review, the changes are shown in blue font in the revised manuscript. We look forward to your decision.

Gregory Lecrivain, on behalf of all co-authors.

Response to Reviewer 1

1.1. It is necessary to address the precise role of such 2d simulations, and this is not done here. Are they to be compared with 2d experiments? I do not see this here, except in very vague form, although such experiments are in the literature.

We have stressed the role of the 2D simulations throughout the manuscript in several ways. In the the last paragraph of the introduction, we rephrased as follows.

> "In the following, we restrict however ourselves to two-dimensional systems and demonstrate the ability of the method to handle collision-dominated cases. Three-dimensional systems will be included in future works"

We clarified the role of the 2D simulations in the method section by bringing forward an analogy to experimental tests available in the literature.

> "The velocity u_x is our attempt to reproduce the quasi-two-dimensional experimental system in References [1, 2]. In such systems, a horizontal mono-layer bed of bubbles is confined between either a column of water and an upper plate or between two plates. Experimentally, the foam velocity is set at the inlet by imposing a constant flow rate."

We also provided a new figure comparing the flow field of the foam to experimental data [2] (See response 1.5).

1.2. For a time, experiments with a rotating rheometer, raising similar issues, were particularly in vogue. These were considered interesting in as much as they showed the phenomenon of shear localisation (related to the phenomenon of "stagnancy" in the present paper) - its interpretation was debated. The discussion of those findings highlighted an important difference between two and three dimensions in that boundary conditions that confined the system to two dimensions (sometimes involving two glass plates) played a role that is entirely absent in 3d.

We are well aware of this discussion regarding shear banding. We have added this point to the discussion of Figure 10.

> "The obstacle can be regarded as an artificial wall roughness. Consequently, the bubbles at the wall would remain immobile while those near the channel centerline would move faster. However, as can be seen in Figures 6-7, this did not occur. We had hoped to observe an accumulation of T1 events along some horizontal lines. The spatial distribution of the T1 events does however not point to any shear localization or shear banding [3]. Figure 10 shows that the T1 events occur mostly up- and downstream of the obstacle, not supporting the prediction of shear banding. Three-dimensional simulations and an appropriate subgrid wall boundary condition would probably be necessary to further investigate shear banding."

1.3. Very little of this background is properly acknowledged/incorporated/integrated in the present paper. Another surprising instance of this shortcoming is the lack of any mention of constitutive relations (although there is some related discussion late in the paper) is some : whatever their merits, they have played a large role in past debates and analyses.

Our introduction is designed to reflect the lack of 3D measurement techniques and the state of the art regarding numerical simulations of flowing foam to compensate for that. To each sentence in the introduction, we added a short description briefly explaining the major findings. One example is given below.

> "The vertex model, where the dry foam is represented by a set of polygons, has been used to study foam shearing [4, 5]. In the latter work, two-dimensional simulations of foam shearing in an unobstructed channel were performed. The authors investigated the foam stress in the non-quasi static regime and found that the number of topological rearrangements, so-called T1-events, correlated with bubble elongation."

Regarding the constitutive relation, this would be definitely interesting. In particular with regard to the wide range of liquid fractions that can be simulated with phase field models. Unfortunately, we cannot contribute to these discussions yet, as our simulations do not yield a force to drive the flow. Consequently, we cannot measure the complex shear modulus of the foam.

1.4. It should be made clear whether the 2d system is supposed to correspond to a real one, or is just a temporary stand-in for a 3d one.

We fully agree with the referee that such 2d simulations and experiments of foam flow through a confinement or around a cylinder are well investigated both experimentally and numerically. However, one key ingredient in our work is significantly different, that is the liquid fraction. Experiments are typically performed at much lower liquid fractions. The key feature of our technique is, that we can investigate liquid fractions from bubbly liquid down to dry foam with the same model without tuning or switching any parameter. This allows in our simulations at liquid fractions near the jamming point to have jammed bubbles before the obstacle and mobile bubbles behind it. To the best of our knowledge, this has not yet been investigated. So ultimately, our technique should be able to compute the complete foam generation process from rising bubbles to dry foam.

> "Compared to other foam simulation tools cited in the introduction, the present method has one major advantage. It is namely the capability to consider bubbly flow, wet and even dry foams coexisting in one single simulation. This allows to investigate in particular the flowing behavior of wet foam near and across the jamming limit."

The model itself is applicable to three dimensional systems. However, this requires significant extension of the code, which might take several months and we are currently not sure if we get the funding for that. Consequently, with this work we want to show the potential of phase-field methods in foam flow and make use of the key feature of variable liquid fractions to shed light on the un-jamming behind the obstacle at intermediate liquid fractions. This has been clarified in the conclusions.

> "Future work will also include an extension to three-dimensional simulations. This can be easily achieved with the phase-field model, because in contrast to other methods, e.g. Volume-of-Fluid methods, the transport equation for the scalar identification functions do not differ substantially between two and three dimensions"

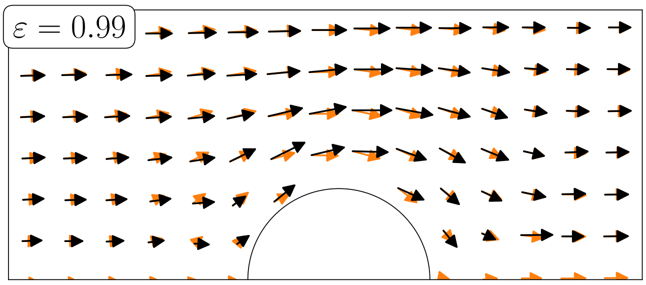

Figure 18: Foam velocity vectors (black) overlaid with those obtained experimentally (orange) [2]. The foam is mono-disperse in the experiment and simulation. ε is the gas fraction.

1.5. It may be helpful if more of the technical material is relegated to appendices or supplementary material. Ideally, definite comparisons with experiments (not necessarily very detailed) should be made, either using data in the literature

We added a new Appendix B, where the time-average flow field is compared to experimental data from the literature.

> "We compare the time-averaged velocity field of the driest mono-disperse foam to experimental data. In the original experiment [2], a mono-layer bed of bubbles is confined between a lower liquid pool and an upper transparent wall. A cylinder is placed in the channel center and the bubbles flow around it. The experimental scenario is somewhat different to that presently simulated. Here, two half-disks are located at the wall. In the numerical and experimental scenarios, the ratios of the equivalent bubble to obstacle diameter have the same magnitude order and equate $d_b/(2R) = 0.37$ and 0.15, respectively. Figure 18 compares the numerical and experimental velocity vectors, that have been digitized from the publication. Despite the differences between the two systems in terms of obstacle position, the magnitude and the direction of the foam velocity qualitatively match. The discrepancies tend to increase as one approaches the obstacle."

1.6. The general nature of the phase field method should be more clearly expounded.

We added a brief and general introduction to phase field modeling before narrowing down to the model description. Because the manuscript is already quite long, we refrained from a larger text extension. We added a review article [6] for further reading on modeling of multi-component flows with phase fields.

> "The general idea behind the phase field modeling is to introduce a dimensionless identification function, also called order parameter, that continuously varies over a thin interfacial region. The sharp fluid interface between the gas and liquid phases is hence smeared out with a thin but nonzero transitional region. We refer the reader to the review article [6] for further information on the phase-field modeling of multi-component flows."

1.7. Foam compression is a rather ambiguous term, that needs clarification. A foam can be compressed in two different ways: simply change the volume, or change the liquid fraction.

We have clarified this point.

> "Compression refers here to the local change in liquid fraction."

1.8. The description of the Surface Evolver (which may be unnecessary) is obscure.

We think, the Surface Evolver is an important numerical tool to mention here. We have refined the description as follows.

> "Large advances in this field have been achieved with the Surface Evolver [7], which discretizes the air-liquid interfaces of the foam with a triangle mesh. Under a bubble volume conservation constraint, it then iteratively redistributes the triangle nodes to find the energy minima in terms of foam shape."

1.9 Some of the details of figures are hard to discern and perhaps should be enlarged.
We have enlarged the fonts in all figures.

Response to Reviewer 2

2.1. The advection velocity shown in (8) seems a bit odd for this system, since it simply represents a flat velocity profile in the whole channel. This is not a velocity field you would expect for the flow through a constriction, since the velocity would be higher in the constriction due to mass conservation. Could the authors elaborate a bit more on the choice of this advection velocity? Note that the bubble velocity indeed increases in the constriction, due to the area-conservation constraint.
We added the following clarification in the manuscript.

> "We here decide to drive the flow by shifting all bubbles with a small incremental distance ($u_x \delta t$) over the duration of one time step (δt). This is here achieved by prescribing the constant advection velocity u_x in Equation 8. The velocity u_x is our attempt to reproduce the quasi-two-dimensional experimental system in References [2, 1]. In such system, a horizontal mono-layer bed of bubbles is confined between either a column of water and an upper plate or between two plates. Experimentally, the foam velocity is set at the inlet by imposing a constant flow rate. In reality, the Eulerian velocity $\boldsymbol{u}$ should vary in space. That is however not the case in the presented simulations. In the constriction for example, that is at $x = 0$, $\boldsymbol{u}$ should increase because of mass conservation and at the wall, it should be zero. A constant u_x might rather be reminiscent of the flotation process [8], where swarms of gas bubbles rise in a liquid tank with a constant terminal velocity because of buoyancy. Nonetheless, this first work is an alternative and promising method for future foam simulations."

2.2. Since the focus is on the physics of the problem, not many details are given on the numerical method are given. For example, was it verified that the results are independent of the mesh-size and time-step size?
We have preformed an additional mesh convergence study. The results are presented the new Appendix A, which reads as follows.

> "A grid dependency test is here presented. Five Cartesian grids discretizing the channel into $N_x = 112, 144, 176, 208, 240$ and $N_y = 70, 90, 110, 130, 150$ nodes are considered, where N_x and N_y are the number of nodes in the axial and transverse directions of the channel flow. In all simulations, the ratio of the sharp bubble diameter to the channel height is kept constant and corresponds to the mono-disperse scenario $\varepsilon = 0.68$ illustrated in Figure 5, that is $\bar{d}/H = 0.12$. The interfacial width is set to the size of one grid element, that is $\xi = \Delta$, and is the only nonconstant value. The time-average velocity along the channel center line $V_c(x)$ is determined for the fives grids. As seen in Figure 17, all simulations deliver similar velocity profiles. In this work, simulation data obtained with the finest grid, that is $N_x = 150$ and $N_y = 240$, are discussed."

In terms of time-step size, we have provided the Courant number.

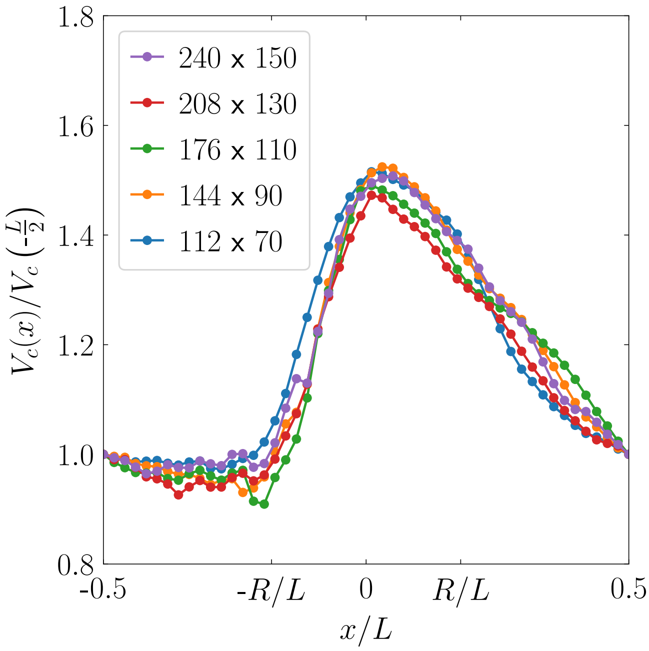

Figure 17: Grid dependency test. Time-average velocity along the channel center line $V_c(x)$ with increasing mesh density.

> "The time step δt is calculated from the Courant number here set to $Co = (u_x \delta t)/\Delta$ = 10^{-4}."

2.3. Typo: beeing in the caption of Fig. 9

Thank you for pointing this out. We corrected the caption.

References

[1] B. Dollet, A. Scagliarini, and M. Sbragaglia. "Two-dimensional plastic flow of foams and emulsions in a channel: experiments and Lattice Boltzmann simulations". In: *Journal of Fluid Mechanics* 766 (2015), pp. 556–589. DOI: 10.1017/jfm.2015.28.

[2] Benjamin Dollet and François Graner. "Two-dimensional flow of foam around a circular obstacle: local measurements of elasticity, plasticity and flow". In: *Journal of Fluid Mechanics* 585 (2007), pp. 181–211. DOI: 10.1017/s0022112007006830.

[3] Thibaut Divoux et al. "Shear Banding of Complex Fluids". In: *Annual Review of Fluid Mechanics* 48.1 (2016), pp. 81–103. DOI: 10.1146/annurev-fluid-122414-034416. eprint: https://doi.org/10.1146/annurev-fluid-122414-034416. URL: https://doi.org/10.1146/annurev-fluid-122414-034416.

[4] I. Cantat. "Gibbs elasticity effect in foam shear flows: a non quasi-static 2D numerical simulation". In: *Soft Matter* 7 (2 2011), pp. 448–455. DOI: 10.1039/C0SM00657B. URL: http://dx.doi.org/10.1039/C0SM00657B.

[5] N. Kern et al. "Two-dimensional viscous froth model for foam dynamics". In: *Physical Review E* 70 (4 2004), p. 041411. DOI: 10.1103/PhysRevE.70.041411. URL: https://link.aps.org/doi/10.1103/PhysRevE.70.041411.

[6] Junseok Kim. "Phase-Field Models for Multi-Component Fluid Flows". In: *Communications in Computational Physics* 12.3 (2012), pp. 613–661. DOI: 10.4208/cicp.301110.040811a.

[7] Kenneth A Brakke. “The Surface Evolver”. In: *Experimental Mathematics* 1.2 (1992), pp. 141–165.

[8] G. Lecrivain et al. “Attachment of solid elongated particles on the surface of a stationary gas bubble”. In: *International Journal of Multiphase Flow* 71 (2015), pp. 83–93.