I find the manuscript entitled “Modelling chemical advection during magma ascent” by Dominguez et al. quite interesting and helpful for the geoscience community. In my view, the problem of numerical advection is often underappreciated, and the presented study clearly shows the pitfalls of using too simplistic advection schemes. In their work, the authors introduce and compare the performance of four selected numerical advection schemes. In conclusion, they advocate the usage of the WENO type of schemes for petrological applications (chemical advection). Below I give my general and detailed suggestions that the authors may consider to follow before the final publication of the manuscript.

The numerical schemes selected for the study are introduced as simplified 1D variants and later tested using 2D setups. Although the 2D generalizations are rather straight-forward, in my view, it would be worth to give some clues to the readers on how they are developed. In particular, the 2D generalization of the WENO scheme that builds on a 5-point stencil in 1D is perhaps least obvious in this respect (not to mention the treatment of the boundary conditions). In fact, in the final parts of the manuscript, the authors themselves mention an alternative formulation for the 2D WENO scheme that builds on a 3x3 stencil.

I was a bit confused about the MIC scheme that was used in the study. The authors mention the initial step of interpolating the field values from grid nodes to markers. Considerable focus was given to integrating the trajectories of the markers. The step of interpolating the field values back from markers to nodes was sketched out. Unless I got it wrong, the advected fields in this study are essentially phase fields, so no calculations and updates need to be performed on the Eulerian grid. Thus, there is no need to interpolate back these updates to markers in such cases. However, interpolating the base field values from grid nodes to markers would be an undesired and suboptimal step leading to unnecessary numerical diffusion. I think that it would be helpful to clarify the design of the MIC scheme that was used in the tests. What is the reason for the observed mass loss in the second test using the MIC scheme? Could it be related to the non-conservative reseeding and removal of the markers? I would suspect that the mass conservation of the standard MIC should be almost ideal.

The author presented the rigid-body rotation test as their base test. In my view, a bit more complex synthetic 2D test such as for example the shear cell test would be a great addition to the study. On a different note, I actually wonder how to design a balanced comparison of the accuracy of grid-based advection schemes against marker-based methods, when their numerical resolutions largely differ (for a fixed grid resolution, a much larger number of markers is used than nodes in this study). In addition, the authors took advantage of the extended stability of the MIC and SL schemes and used a larger value of the time step than in the case of the upwind and WENO schemes, however, this may have an effect on the numerical performance of the studied schemes.

I was confused by the fact that it was not possible to obtain MIC results using a 128 GB machine for grid resolutions such as 500x1000, when the number of markers is on the order of 10 million. What was the exact reason for such limit in the current MIC implementation? The final conclusions build on the statement that the MIC scheme lacks mass conservation and it is expensive in terms of computation and memory. I already expressed my concerns about the observed lack of mass conservation in the case of the MIC scheme. I think that it would be also

useful if the authors could elaborate more on the computational and memory performance of their MIC implementation.

Detailed comments:

l. 38 What is exactly the notion of triviality in this case?

l. 43 “This brings limitation to the resolution of the model …” - It is indeed correct that for any given computational resources 1D numerical models can be studied with a higher resolution than 2D models. On the other hand, looking from a 3D physical perspective using approximate 2D numerical models could be considered as relaxing computational limitations to the resolution of the model.

eq.5 Is it necessary to use the brackets around C_e^f?

l. 84 Perhaps early rather than earlier?

l. 87 “It consists at tracking individual particles on a Lagrangian frame and to reinterpolate them when needed on an Eulerian stationary mesh grid”. consist at or consist of? to reinterpolate or reinterpolating?

l. 94 “..to the position of a fixed Eulerian grid..” It would be perhaps worth to mention grid nodes here.

l. 106 Please change spacial to spatial.

l. 106 It might be a bit confusing when the term “element” is mentioned here.

eq. 6 It would be perhaps more transparent if the f symbol, which stands for fluid, was used in the superscript rather than subscript. It could be mentioned that v denotes the x component of the velocity.

l. 109 Please consider using “grid spacing” rather than “increment in space”.

eq. 9 Please consider using the f symbol in the superscript. The j index that denotes the spatial component of the velocity could be then used in the subscript.

l. 129 Here and in fig. 1 caption, “Upwind” is capitalized, while it is not in l. 111.

l. 129 Please consider removing the comma.

l. 133 Perhaps “for a single grid node” rather than “1 element” would suit better here. This issue reappears in other parts of the manuscript.

l. 153-54: What is meant by “complex problems” here? I suspect that for any problem boundary conditions need a special or careful treatment when a 5-point stencil is used. I don’t

think that it is necessary to present in details how BC are treated in this case, but a short description would be useful.

l. 158 It is not that important, but I'm wondering about the notion of "fully capture" in this case.

l. 164 Please replace spacial by spatial.

l. 171 What is "therefore" referring to?

l. 182 " … is not easy to determine xd. A common approach to overcome this limit…" In my vewi, the term limit may be misleading in this context. I would suggest something like: A common approach to accurately determine …

l. 193 "a stencil point on the grid" Perhaps it could be described as "a grid node".

l. 196 It could be several (neighboring) cells depending on the interpolation order (here cubic, so four nodes are needed to span the interpolation space). On the other hand, it depends on the adopted definition of the cell. Anyway, I think that it would be worth to clarify this issue.

l. 214 "…interpolate their values…". Perhaps "…interpolate field values associated with them…"

l. 220 "The initial value of each marker …" Perhaps "The initial field value in each marker …"

l. 222-23 The problem of deteriorating marker resolution is not only limited to highly divergent flows. It may also affect incompressible, but strongly stretching flows. In such cases, the density of markers can actually depend on the direction (markers may be locally jammed in a certain direction and rarefied in the direction that is perpendicular). Thus, simple scalar measures of marker density may fail in such cases.

L. 246-54 In my view, based on the presented description, it is not exactly clear how the field values are interpolated from markers to grid nodes. Is it the least-square type of linear interpolation? How many surrounding cells are used?

l. 256-261 The domain size is given as dimensionless, while the time is in seconds.

l. 268 Isn't S_k constant for all the internal cells?

l. 352-354 Is this the description of the ODE solver implemented in DifferentialEquations.jl (l. 348)? Please clarify.

l. 363 The standard deviation is given as a dimensionless quantity, while the size of the model domain is given in meters.

l. 366-67 I might be missing something here, but I don't exactly understand how the initial distribution of the melt chemistry is governed. In fig. 8 it appears just bi-modal at t=0 Ma.

Table 3 Please fix the value of the shear modulus (an incorrect exponent is used).

l. 374-383 This part might be a bit confusing. Although the link to eq. 5 is mentioned in l. 320, I think that it would be useful to clarify which physical fields are advected in this model. Please explain why the oxide content may not be conserved, despite the melt fraction is claimed to be conserved due to the chosen numerical scheme. If the total mass of the melt composition (summed weight percentages?) is renormalized in each time step, then I'm not sure if it could be treated as a conserved quantity.

l. 389 What is meant by mixture here?

l. 397 I am a bit confused by the observed lack of mass conservation in the upwind scheme. Is it due to the fact that the renormalization procedure is used? I would also expect MIC to be essentially mass conserving and non-diffusive. Are the observed mass loss effects due to marker reseeding and removal?

l. 407 Using mid-point for computing trajectories in MIC and SL results in some iterations.

l. 423 What is exactly meant by complex boundary conditions?

Fig. 9 & 10 The axes are logarithmic and there is no need to add log(.) in the labels.