Kyle

在 2018年8月20日星期一 UTC+2下午12:59:35，Quentin Magdelaine写道：
- mostrar texto das mensagens anteriores -

Quentin Magdelaine 31/08/18

Traduzir mensagem para português

Hi Kyle,

1. To use it with a Navier-Stokes solver it rather easy, you have to :
- replace #include "../advection_Q.h" by #include "navier-stokes/centered.h" and #include "tracers.h",
- save the uf field in other field (let's say uf_save) at the beginning of the stability event (you have to declare it as a global field because you will need it later),
- add the evaporation velocity to uf instead to replace it, so apply it to another field (let's say ev) and add ev to uf,
- after the vof advection, in the tracer_advection event, instead of setting uf to 0, you have to set it back uf_save.
To have surface tension, just add #include "two-phase.h" and #include "tension.h" (f[] and interfaces are allocated in "two-phase.h", so you will have to suppress this two allocations in your .c file). For evaporation, it will work (it should, if not tell me), for condensation, it is probably unstable without surface tension, but if you include "tension.h", it may work.

2. As written at the beginning of the page, for evaporation, the Peclet is directly the ration of the saturation mass concentration of the liquid's vapor over the density of the liquid. For water I think it is something around 1e-5, but here I took 1e-3. D is the diffusion coefficient: since it is one of the parameters I use to obtain my non-dimensional variables, it is 1. The saturation concentration of vapor (vcs) is my scale of vapor concentration, so by definition it is 1, and I choose to have 20% of humidity at my boundary, so cinf is 0.2. At last, the dirichlet time factor is not so easy to choose. But it works well for me if it is set to 10, so you can try to keep it as it is (it is not dimensional).

3. See question 1