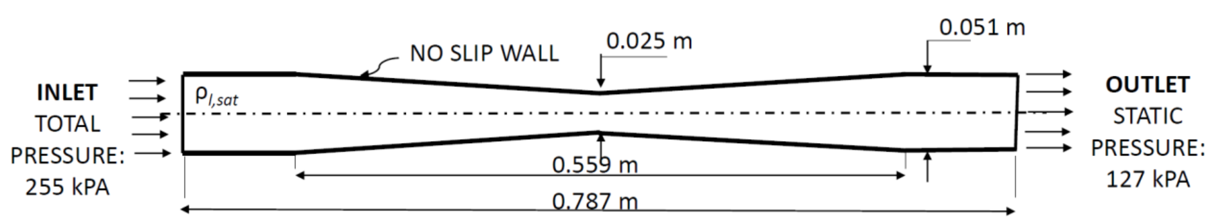
OPENFOAM ISSUE REPORT

PROBLEM DEFINITION

The problem being simulated is flashing (phase change from liquid to gas) of steam in a convergent-divergent nozzle. Pure liquid enters the nozzle at the inlet and the phases change occurs downstream of the throat due the pressure dropping below the saturation value. The solver being used is the *reactingTwoPhaseEulerFoam* with the *interfaceCompositionPhaseChangeTwoPhaseSystem* phase system. The interface composition system being used is the *Saturated* type with both the constant and the antonie type are tested. A single species exists in each of the phase.



PROBLEMS ENCOUNTERED

1. The phase change is triggered only with specification of a small value of the initial volume fraction of the gas phase (1e-12).
2. Unexplained temperature drop is developed and convected near the first control volume near the inlet boundary. This leads to low temperatures being transported throughout the nozzle thereby leading to drop in saturation pressures and temperatures.

BOUNDARY CONDITIONS

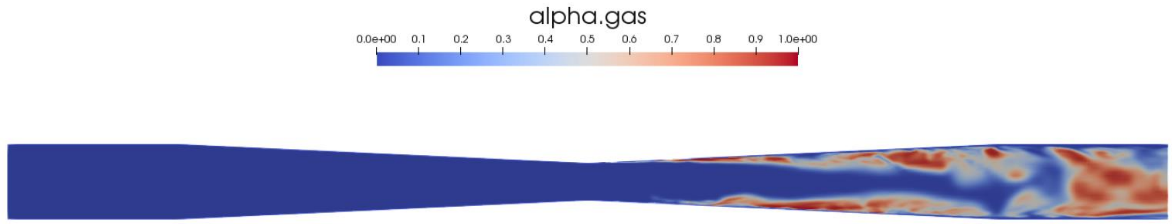
Only the laminar simulation is tested. The most successful boundary conditions are as below:

	INLET	OUTLET	WALL
p_rgh	prghTotalPressure	prghPressure	zeroGradient
p	calculated	calculated	zeroGradient
U.liquid	pressureInletVelocity	pressureInletOutletVelocity	noSlip
U.gas	pressureInletVelocity	pressureInletOutletVelocity	noSlip
alpha.gas	fixedValue	zeroGradient	zeroGradient
alpha.liquid	fixedValue	zeroGradient	zeroGradient
T.gas	fixedValue	inletOutlet	zeroGradient
T.liquid	fixedValue	inletOutlet	zeroGradient
water.gas	zeroGradient	inletOutlet	zeroGradient

water.liquid	zeroGradient	inletOutlet	zeroGradient

RESULTS

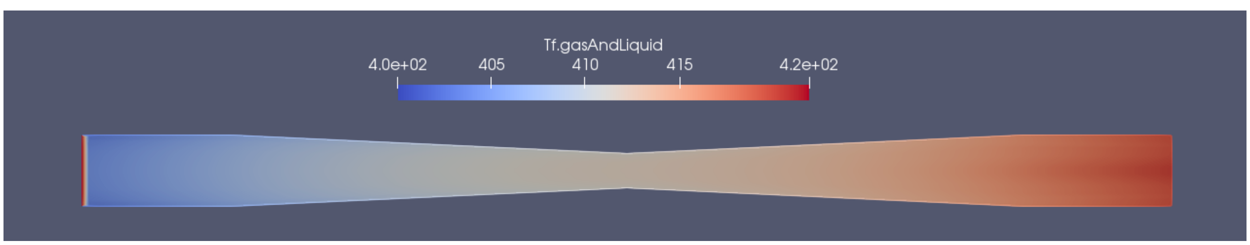
The above setup gave good results only once, which is shown as below:



The case files can be accessed at the below link:

https://www.dropbox.com/s/pejy7nkj2de16ut/CD_NOZ_3D_rtpef_BNL_291_lam_constant_TP_fm_off_algas_correct_upload.tar.xz?dl=1

However, when I try to re-run the exact same setup by deleting the intermittent time steps, I find that low temperatures are developed near the inlet and the interface temperature looks something like this. The inlet temperature is set to 421.9 K and rest of the boundaries are set to zeroGradient.



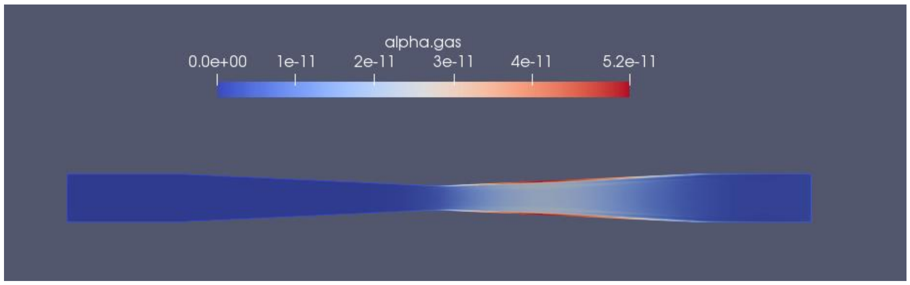
The video file of the simulation can be accessed at

https://www.dropbox.com/s/q4mmce0wgg0i467/t_drp.ogv?dl=1

The case files with a coarser mesh (100k nodes) with the same case setup can be downloaded here for re-run if needed

https://www.dropbox.com/s/5x8ic08kcfu06n7/CD_NOZ_3D_rtpef_BNL_291_lam_cse8_TP_old case.tar.xz?dl=1

This issue does not occur when I use the mapped inlet velocity (no average) but the phase change is completely different and is not sustainable. The same *prghTotalPressure* boundary condition is used but the *pressureInletVelocity* is replaced by the mapped velocity (no averaging) at the inlet.



The setup files for this simulation can be downloaded here:

https://www.dropbox.com/s/c2rzyp8gd2ta5rm/CD_NOZ_3D_rtpef_BNL_291_lam_cse22_mappedvel_noavg_algas_on.tar.xz?dl=1