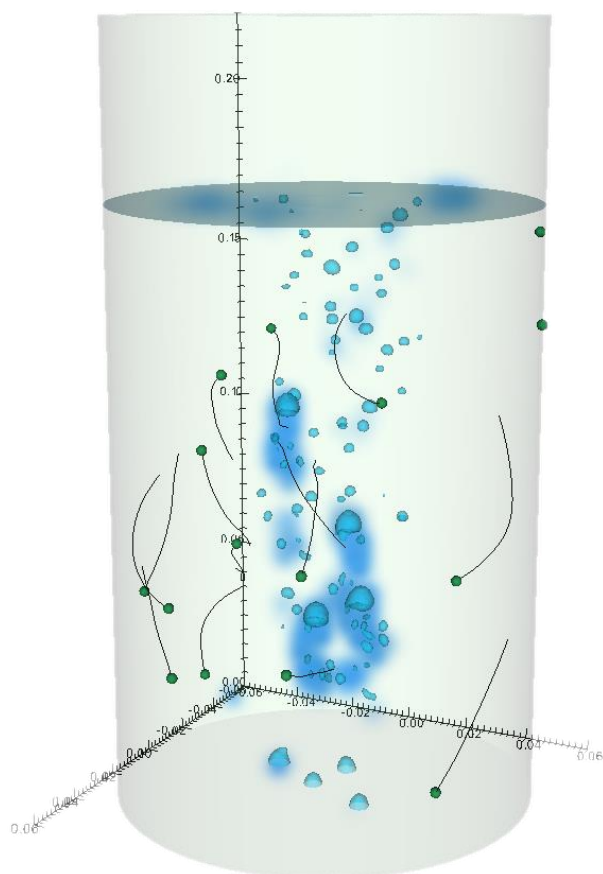
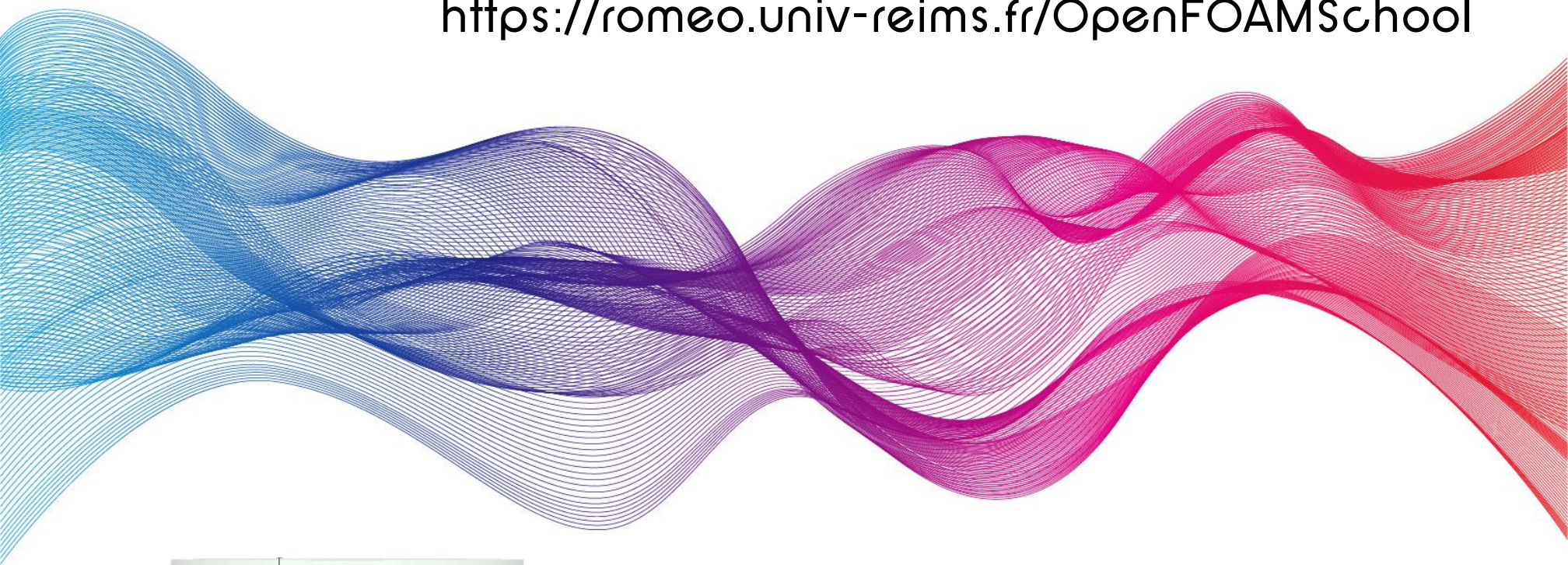


# Open FOAM

## formation

6-8 novembre 2017, Paris-Saclay

Programme / Informations / Inscriptions limitées :  
<https://romeo.univ-reims.fr/OpenFOAMSchool>



Initiation CFD,  
cas pratiques  
OpenFOAM  
et lancement sur  
supercalculateur