WORKSHOP
Direct and Large-Eddy Simulation 12
June 5th-7th 2019, Madrid, Spain

DNS OF TURBULENT CHANNEL FLOW WITH CONDENSATION AND A CLUSTER-BASED DROPLET DEPOSITION MODEL

P. Bahavar[1], **C. Wagner**[1,2]
[1] Institute of Aerodynamics and Flow Technology, German Aerospace Center (DLR), Germany
[2] Institute of Thermodynamics and Fluid Mechanics, Technische Universität Ilmenau, Germany
philipp.bahavar@dlr.de

INTRODUCTION

Heat and mass transfer in flows with a condensable component are important characteristics of systems ranging from large-scale atmospheric flows to microchip cooling systems. Investigating the interplay of the phase transition and the turbulence of the flow remains an important objective for highly resolved numerical simulations [1]. In this work, single phase direct numerical simulations (DNS) of turbulent flow of humid air along a channel are performed and coupled with a simplified condensation model. This approach provides full information on the flow while simultaneously incorporating selected aspects of the phase transition into the calculation. In particular, the release of latent heat and the deposition of droplets at the channel wall during condensation events are included for this investigation.

GOVERNING EQUATIONS

Assuming incompressible fluid,

$$\nabla \cdot \vec{u} = 0, \tag{1}$$

the evolution of fluid velocity and temperature is given by

$$\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla)\vec{u} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \vec{u} - \vec{f} \tag{2}$$

and

$$\frac{\partial T}{\partial t} + \nabla \cdot (\vec{u}T) = \kappa \nabla^2 T + \sigma_T \tag{3}$$

with kinematic viscosity ν and thermal diffusivity κ. Additionally, the condensable component of the fluid is described via the vapor concentration,

$$\frac{\partial c}{\partial t} + \nabla \cdot (\vec{u}c) = D\nabla^2 c - \sigma_c, \tag{4}$$

where D is the mass diffusivity. The force term $\vec{f}$ is given by the buoyant forces due to density changing with both temperature and vapor concentration [2], linearized as per the Boussinesq approximation,

$$\vec{f} = \beta_T (T - T_{ref})\vec{g} + \beta_c (c - c_{ref})\vec{g}. \tag{5}$$

Here, $\beta_T = 1/\rho_{ref}\, \partial\rho/\partial T|_{T_{ref}}$ is the thermal and $\beta_c = 1/\rho_{ref}\, \partial\rho/\partial c|_{c_{ref}}$ the solutal expansion coefficient of the fluid mixture. Phase change effects are included using the source terms

$$\sigma_c = (c - c_{sat}(T))/\Delta t \quad \text{if } c > c_{sat}(T), \text{ else } 0, \tag{6}$$

which describes the instantaneous removal of vapor from the system if oversaturation conditions are met, and $\sigma_T = \sigma_c\, h_v/c_p$, which in turn adds the corresponding amount of latent heat h_v to the fluid with heat capacity c_p. The liquid phase is not simulated, but the amount of accumulating condensate is tracked for each computational cell.

NUMERICAL SETUP

The channel is set up in a vertical configuration, with gravity acting in the direction of the mean flow. Cyclic boundary conditions are employed in the spanwise direction of the channel. The presence of the source terms for the scalar fields suggests a departure from streamwise periodic boundaries for the investigation domain. A biperiodic precursor domain is coupled to the investigation domain, providing a velocity inflow boundary condition consistent with fully developed isothermal turbulent channel flow to the aperiodic domain [3]. Impermeable walls separated by a distance 2δ comprise the channel boundaries. No-slip conditions are imposed on the velocity. An isothermal boundary is set at one of the walls, with wall temperature T_w below the temperature at the inlet T_{in} on the one hand, providing a cooling effect on the flow, and below the saturation temperature at the inlet $T_{sat}(c_{in})$ on the other hand, facilitating the onset of condensation in the forming thermal boundary layer. The opposite wall is kept adiabatic to eliminate additional perturbation of the scalar fields.

The DNS is performed with the finite volume code OpenFOAM, using second order central differences and an explicit leapfrog-Euler time marching scheme [4]. A projection method is used to couple velocity and pressure to satisfy the continuity condition. Both the precursor and the investigation domain are discretized using hexahedral cells with a resolution of $\Delta x^+ = 5.3$ along the streamwise direction, $\Delta z^+ = 0.27$ for the spanwise direction and ranging from $\Delta y^+ = 0.2 - 3.3$ from the walls to the bulk in the wall-normal direction.

Flow with a bulk Reynolds number $\mathrm{Re}_\delta = 2000$ is enforced by a fixed volume flux applied at the inlet. Thermal and mass diffusion are characterized by the Prandtl number $\mathrm{Pr} = 0.76$ and Schmidt number $\mathrm{Sc} = 0.65$, respectively. The thermal and solutal contributions comprise the Grashof number $\mathrm{Gr} = g\delta^3/\nu^2\, (\beta_T \Delta T + \beta_c \Delta c) = 38840$. The ratio of latent

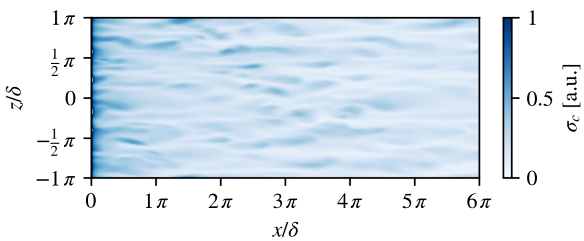

Figure 1: Instantaneous distribution of the vapor source term at the cooled wall. The footprint of the coherent sweeps responsible for wall-normal transport is visible in the pattern of condensation.

heat to the associated temperature change is given by the Jakob number $\mathrm{Ja} = 0.012$. The transport properties are chosen to reflect a system of air with trace amounts of water vapor.

RESULTS

After five viscous time units, the simulation reaches a statistically steady state and temporal averaging is performed until the first and second statistical moments are converged. Additionally, averaging is performed along the periodic spanwise direction of the channel. The average velocity field is analyzed conditioned on the instantaneous local fluctuations of temperature and vapor concentration. Analysis provides insight into the way the turbulent flow structures determine the convective transport of the scalar fields. Humidity is carried towards the cooled wall by coherent sweeps, depositing condensate in longitudinal streaks along the length of the channel (Figure 1). The latent heat released in the phase transition modifies the thermal boundary layer, partially counteracting the cooling effect of the wall.

To investigate the effect of the surface modification due to condensate deposition, the channel wall is deformed in order to imitate the the surface roughness caused by condensate droplets. The agglomeration of the liquid phase deposited along the wall into droplets is modeled preserving information on condensate mass and spatial distribution obtained from the DNS. Density-based spatial clustering for applications with noise (DBSCAN) [5] is used to identify connected regions with significantly increased condensate accumulation at the cooled wall. The condensate mass contained in these clusters is then combined, and a droplet with identical mass is positioned at a location representative of the original region. As a first approximation, hemispherical droplets are shown in Figure 2, but more sophisticated shapes determined by the interface energies and gravity will be used in the future for a more accurate representation of the physical system. The deformation of the surface is diffused into the volume mesh using radial basis functions to ensure mesh quality and resolution requirements.

CONCLUSION

Single-phase DNS was performed for cooled turbulent channel flow of a mixture of water vapor and air. Condensation was incorporated into the simulation by introducing source terms into the transport equations for temperature and vapor concentration, which were determined by a local oversaturation condition. Using this approach, the interaction of selected aspects of the phase transition with the turbulent flow could be investigated in a macroscopic system without incurring the high computational costs of a multiphase simulation.

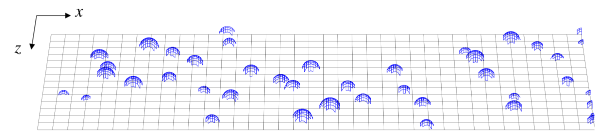

Figure 2: Hemispherical droplets at locations representing regions of high condensate accumulation along the cooled channel wall. Droplet radii are not to scale.

The coherent structures present in the turbulent flow could be shown to influence the spatial distribution of condensation events at the cooled wall. This information was then used by a clustering algorithm to calculate the mass and position of equivalent droplets, which were then modeled by deforming the channel wall to simulate the changed boundary conditions due to condensate droplet agglomeration.

At the conference, further results concerning the interaction of the turbulent flow structures with these modeled droplets and the resulting modifications of heat and mass transfer along the channel will be presented.

REFERENCES

[1] Kharangate, C.R. and Mudawar, I. : Review of computational studies on boiling and condensation, *Int. J. Heat Mass Transfer*, **108**, 1164–1196 (2017)

[2] Bird, R.B., Steward, W.E., Lightfoot, E.N. and Klingenberg, D:J. : Introductory Transport Phenomena, *Wiley New York* (2015)

[3] Bellec, M., Toutant, A. and Olalde, G. : Large Eddy Simulations of thermal boundary layer developments in a turbulent channel flow under asymmetrical heating, *Computers & Fluids*, **151**, 159–176 (2017)

[4] Kath, C. and Wagner, C. : Highly resolved simulations of turbulent mixed convection in a vertical plane channel, *New Results in Numerical and Experimental Fluid Mechanics*, **10**, 515–524 (2016)

[5] Pedregosa, F., Varoquaux, G., Gramfort, A., Michel, V., Thirion, B., Grisel, O., Blondel, M., Prettenhofer, P., Weiss, R., Dubourg, V., Vanderplas, J., Passos, A., Cournapeau, D., Brucher, M., Perrot, M. and Duchesnay, E. : Scikit-learn: Machine Learning in Python, *Journal of Machine Learning Research*, **12**, 2825–2830 (2011)