Abstract Submitted
for the DFD19 Meeting of
The American Physical Society

Direct numerical simulations of a swirling flow in a conical diffuser[1] ANKIT AWASTHI, UGO PIOMELLI, Queen's University — We have performed Direct Numerical Simulations of a swirling flow in a conical diffuser using Nek5000, a spectral-element code. This configuration is a model of the draft tube of a hydroelectric power plant, which is used to increase the flow pressure downstream of the turbine. Separation may adversely impact the performance of the diffuser and should be avoided. A conical diffuser with an opening angle of 20 degrees is chosen. Previous experimental and numerical studies can be used for validation [Clausen et al., *Exp. Therm. Fluid Sci.*, 6(1):39–48, 1993]. The experiment uses a very short inlet section so that at the beginning of the diffuser the flow is not fully developed. The flow is extremely sensitive to the inlet boundary conditions. When the inlet boundary condition for axial and circumferential velocities at the beginning of the diffuser are taken from the experimental study, good agreement is achieved downstream. The use of fully developed pipe flow, on the other hand, results in a very different flow field. The effects of synthetic perturbation and forcing techniques to generate inflow conditions will be described. Future work will include the effect of rough walls on the separation characteristics and pressure recovery.

[1]NSERC and Hydro Quebec

Ankit Awasthi
Queen's University

Date submitted: 22 Jul 2019 Electronic form version 1.4