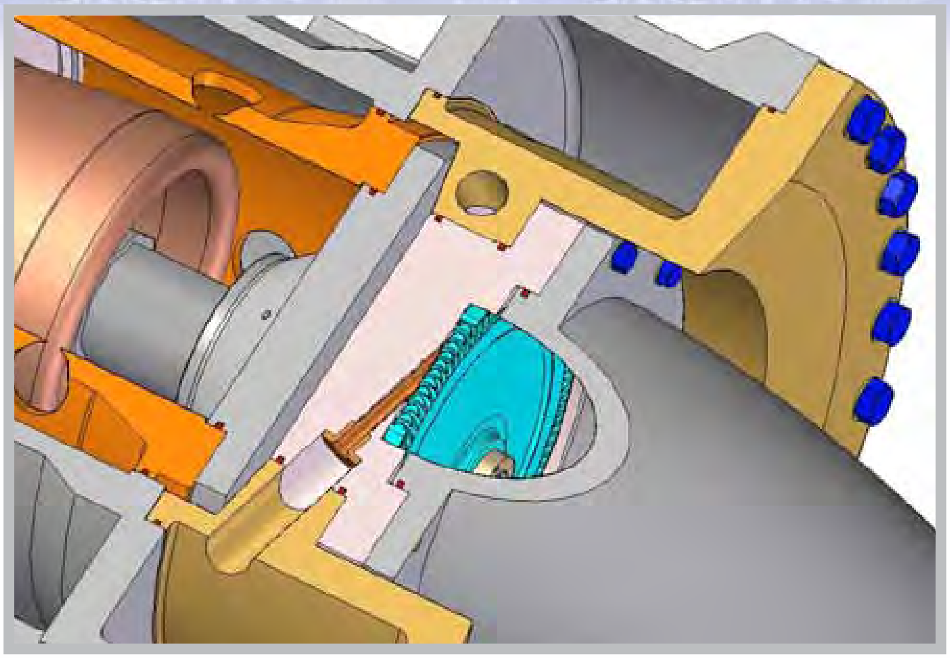
Energent's Variable Phase Turbine Nozzle Flow



Two-Phase Nozzle Flow

Energent's Variable Phase Turbine[1, 2] (VPT) has a set of discrete nozzles upstream of the turbine rotor. The fluid is liquid at the inlet, flashes inside the nozzle, and is two-phase inside the rotor. A previous article[3] discussed calculating the trajectories of droplets inside the turbine rotor. This article will focus on the nozzle.

Energent has been using a 1-D code to design the nozzle. We are doing an experimental program to investigate the two phase flow inside the nozzle. The nozzle will be instrumented to measure pressure and temperature along the flow direction. The momentum flux will be measured indirectly as a contribution to thrust.

In the converging section of the nozzle, the pressure decreases. When it declines to the saturation pressure, vapor bubbles form. At this pressure, the liquid is the continuous phase, the vapor the dispersed phase. With a continued decrease in pressure, eventually the liquid is the dispersed phase as droplets. The development of the dispersed phase, from the formation of vapor bubbles to the transition to liquid droplets and the droplet breakup, is not an easy task to model in the Computational Fluid Dynamics (CFD).

A 3-D CFD study has been started with a commercial code from Numeca. The CFD code is being used under the assumptions of homogenous flow, with zero velocity slip, and the same pressure and temperature on both sides of the liquid-vapor interface. The complexities of the dispersed phase are being ignored.

Figures 1-2 compare the pressure and temperature calculated by the 1-D code and the 3-D CFD code. It is not surprising that they are not quite the same. For the CFD calculation, the two-phase flow was assumed to be homogeneous.

In addition, the flow field around a single droplet is being examined in detail using Direct Numerical Simulation (DNS). DNS solves the equations directly, without

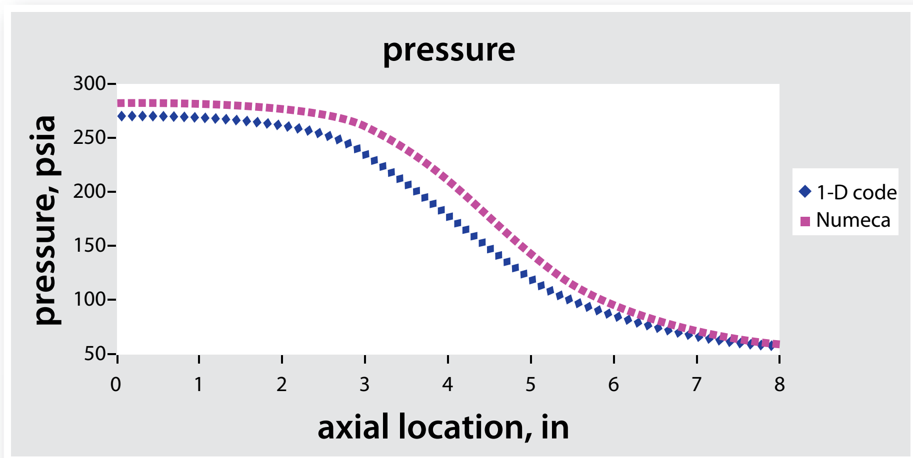


Figure 1. Pressure profile.

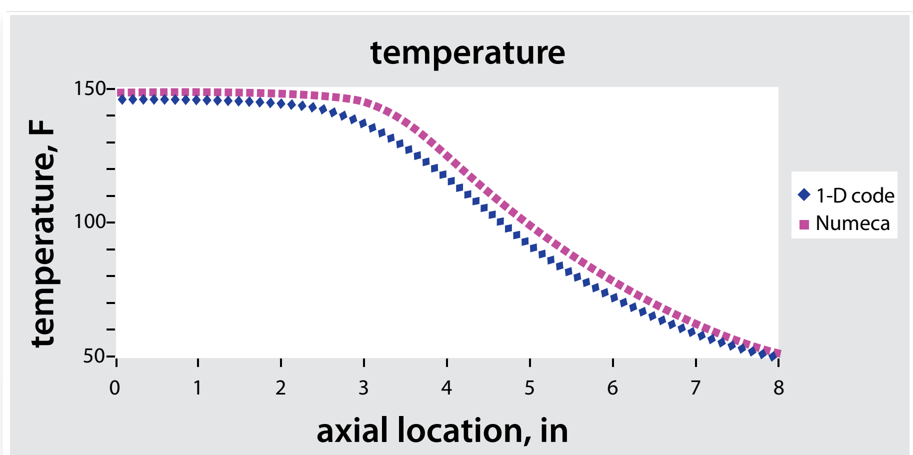


Figure 2. Temperature profile.

needing to employ assumptions in order to model turbulence.

Initially a liquid column is being considered subject to an incoming shock. Figure 3 is a series of snapshots of a column breaking up, displayed as a Schlieren image of the density gradient.

Next the shock will be "smoothed" to better represent the pressure variation from the changing cross-section area inside the nozzle. The information gained from these simulations will be used to develop a reduced order model that can be incorporated into traditional CFD codes as well as the in-house 1-D code.

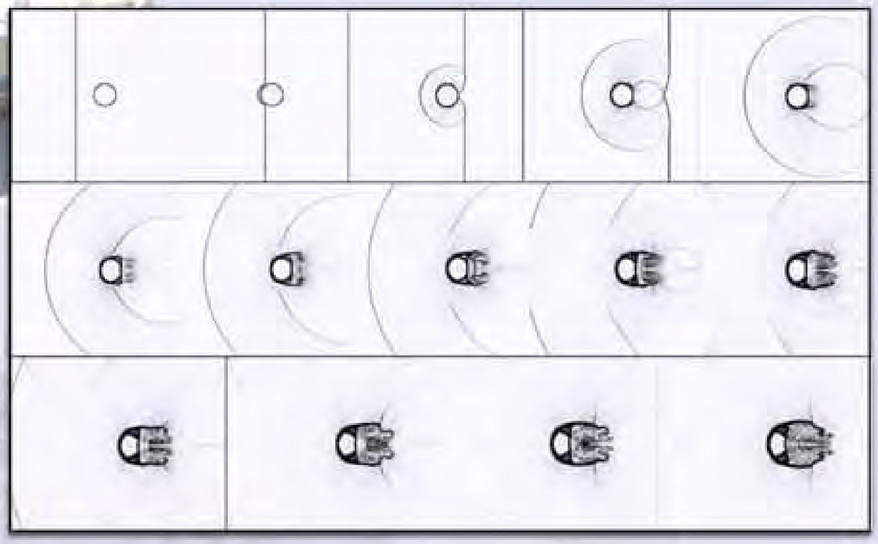


Figure 3. Schlieren images of a shock wave impinging on a liquid column.

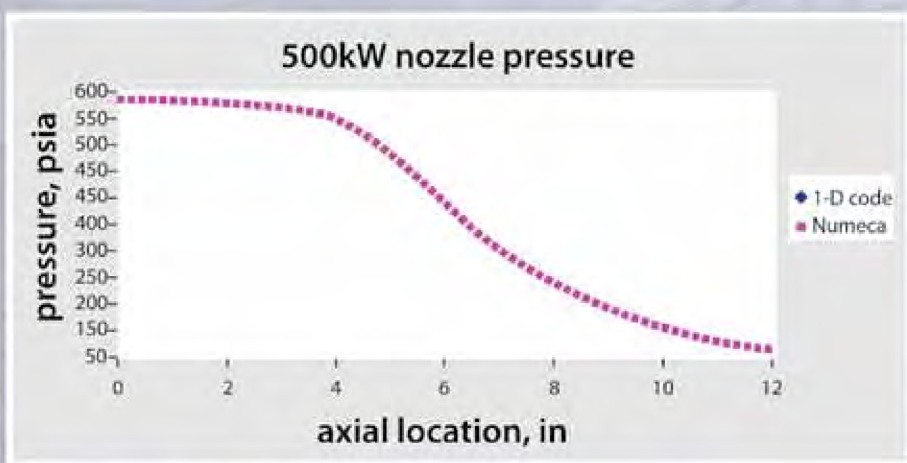


Figure 4. Pressure profile of a 500kW nozzle.

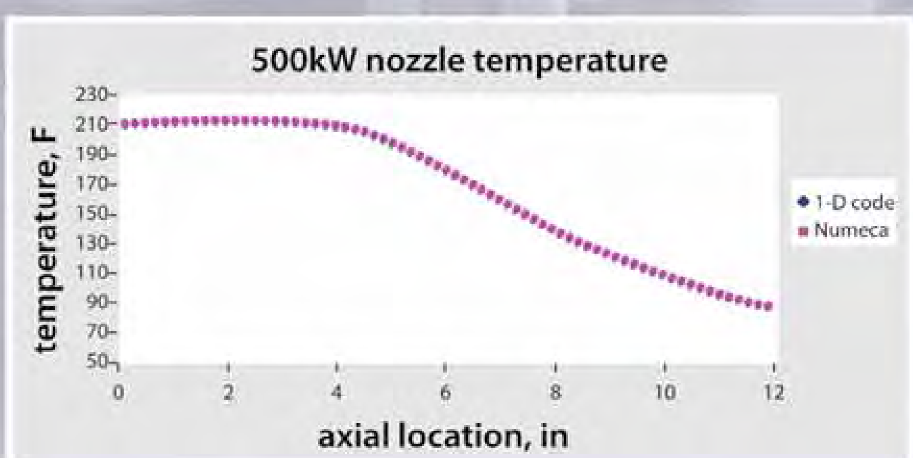


Figure 5. Temperature profile of a 500kW nozzle

When the experiment is completed, a comparison will be made between the numerical calculations and the experimental results.

For a different nozzle, Figures 4-5 compare the pressure and temperature calculated by the 1-D code and the 3-D CFD code. For this nozzle the two codes give similar results. An objective of the two-phase nozzle program is to investigate why the numerical calculations do not always give comparable results.

For more information contact Ron Franz at Energent, 949-261-7533 or rfranz@energent.net.

1 Lance Hays, "The Energent Variable Phase Turbine expands liquids or supercritical fluids used in refrigeration," FrostByte, Summer 2008, pages 1, 4.

2 Lance Hays, "Cryogenic Liquid Expanders," FrostByte, Summer 2011, page 2.

3 Ron Franz, "Droplet Trajectories in a Turbine Rotor," FrostByte, Winter 2011, page 7.