CFD can accurately predict flow behaviors in vessels and equipment and is instrumental in optimizing flow patterns, flow rates, pressure drop, mixing etc. In the following example, CFD is used to simulate the amount of air flow rate that a steam driven ejector nozzle can induce in a fired heater stack. The simulation is used to optimize the flow patterns, nozzle size, ejector efficiency, steam flow rate, and induced air purge rate. Critical to the study also, the simulation provides insight regarding the maximum amount of pressure drop the design can overcome.

Steam passes through the nozzle throat at choked conditions and accelerates as it expands into the stack. This creates a low-pressure zone immediately downstream of the nozzle throat, which entrains air and induces the draft. The ANSYS Fluent powered CFD analysis captures the full effects of multispecies transport, compressibility, high Mach Number velocities, and significant wall boundary interactions. JIS uses ANSYS Fluent because coupling these applications in a single simulation cannot be conducted in many of the other commercially available CFD packages.

**Physics Used in this Study:**
Multispecies Transport, Compressibility, high Mach Number Velocities, Significant Wall Boundary Interactions (mesh sensitivity analysis)
Simulation Results Predict Accurate Flow Patterns
Bulk air streamlines rising up the stack are seen being drawn into the low-pressure region immediately downstream of the nozzle throat.

Flow Streamlines Colored by Mass Fraction of Steam
(mass fraction)
Mesh Sensitivity Analysis

Flow predictions in the nozzle throat and immediately downstream are very sensitive and dependent on the mesh quality. Adequate wall boundary mesh refinement, and mesh smoothing parameters are iterated on to achieve a mesh-independent solution (the solution no longer changes as mesh refinement increases). Computational run time and demand increases with mesh size, so to keep computational demand and cost down, the mesh is optimized in this manner.