Optimising Mixing in a Fuel Injector Using CFD

CFD simulation was used to investigate the performance and then steer the design of a Fuel Injector assembly for Northvale Korting - a leading supplier of valves, eductors and ejectors. Using ANSYS FLUENT, a short design optimisation study was carried out on the baseline product, which helped converge on a solution which met design criteria, while ensuring no other performance parameters were sacrificed.

Company

Since 1903, Northvale Korting has been supplying valves, eductors, ejectors and ancillary equipment to a global clientele in industries spanning defence, energy, fire, food, HVAC, petrochemical and utilities, to name a few. Being actively involved in design research and having a corporate commitment to engineering excellence has meant that Northvale Korting is not just a respected leader in the market, but also marks the required standard in their field.

Background

Wilde Analysis were commissioned to assist in the design of a fuel injector assembly. One of the primary specifications was that there should be no fuel-rich pockets at 13 diameters downstream of the ejector flange i.e. the variation in concentration of the support fuel vapour should be between ±10% of the mean, across the pipe diameter, at this downstream length.

> “” Without the **modelling capabilities and the advice/service** provided by Wilde we would not have been able to identify the limitations in our current design in house nor have been able to implement the required design changes that the CFD process identified as being required.

The basic workings of the Fuel Injector are as follows:

(i) Process gas enters upstream of the injectors
(ii) 99% Methanol of varying droplet sizes are injected via 24 circumferential cone-sprays
(iii) Methanol evaporates based on convective mass transfer or boiling-off at operating pressure
(iv) Evaporated methanol mixes with process gas downstream of injector venture.

Challenge

To accurately capture potential fuel-rich pockets in the downstream section, the simulation had to account for mass transfer via fuel droplet break-up and evaporation.

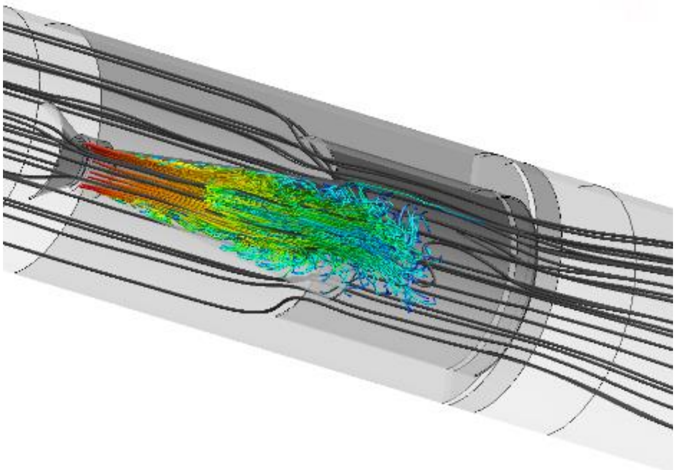

Fig. 1: Fuel Droplet & Process Gas Trajectories

Furthermore, given that there were two fluids within the assembly, the model had to be treated as a multiphase simulation. In order to capture the complex physics accurately, while also assessing multiple design options within permitted timescales required not just an experienced

Fig 2: Coander Effect in Downstream Plume

analyst, but also a software tool that provided key advantages from geometry generation through to the actual solve.

After a short proof-of-capability study across the **ANSYS CFD** suite, **ANSYS FLUENT** was utilised to model the fuel droplet injection, mass transfer and downstream mixing. Using **ANSYS FLUENT** also meant that the 'nozzle state' of the injectors is automatically taken into account when being discharged into the process gas flow. Implicitly, this meant that the internal details of the injector did not need to be modelled and the simulation domain was cut-off at the entry point of the fuel droplets; over a large number of simulations and design iterations, this saves computational time and resources.

Another advantage of using **ANSYS FLUENT** is its ability to model thin surfaces. This capability became particular important when Wilde suggested the use of mixer tabs to provide better mixing and thereby achieve the 13 diameter concentration variation specification.

> "" Wilde was instrumental in the modifying of our existing design to ensure that **performance met the very specific and stringent customer requirements** on this particular project

Although the geometry is symmetric, a full model was utilized for the analyses as initial tests showed the mixed plume did not hold any symmetry down the length of the pipe. As shown in the image below, there is also a tendency for the plume to drift off-centre; this is most likely due to the Coander effect, which again is accounted for in the software.

Results from the baseline design suggested that at 13 diameters downstream of the ejector flange, there were still fuel rich pockets present, for some operating conditions. Consequently, a design that utilized guide vanes was considered. Initially, three rows of vanes were introduced to the system: one at the exit of the injection stream and two at the ejector flange.

The introduction of these additional vanes however posed a new problem, in that these vanes forced a momentum loss and dynamic pressure drop in the fluid streams.

Solution

Consequently a short feasibility study was undertaken to optimize the location, number and relative angle of the guide vanes. The study found that just two rows of vanes in a radially circumferentially staggered arrangement provided a solution that met the design criteria, while also providing minimal momentum loss.

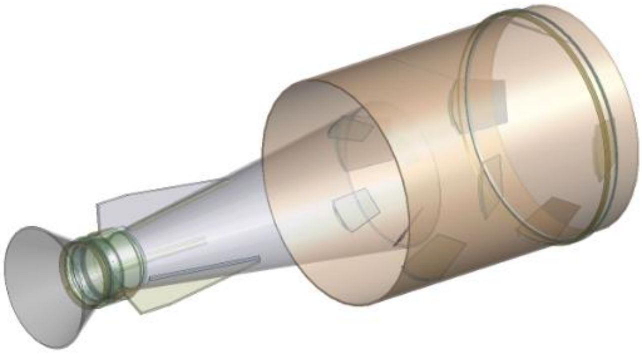

Fig 3: Diametric Variations in Fuel (baseline design on left, optimized design on right)

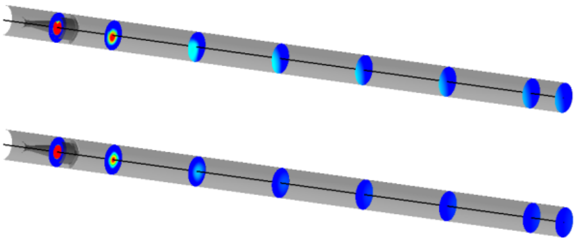

Fig 4: Final Ejector Design with Guide Vanes

Business Benefits

Riding on the success of this initial project, further CFD analyses for Northvale Korting have been undertaken by Wilde. Having taken an active role in the simulations and seeing first-hand the benefits of using CFD, Northvale Korting decided to bring ANSYS CFD & FEA capabilities in-house. The relevant training and software modules were provided by Wilde Analysis directly.