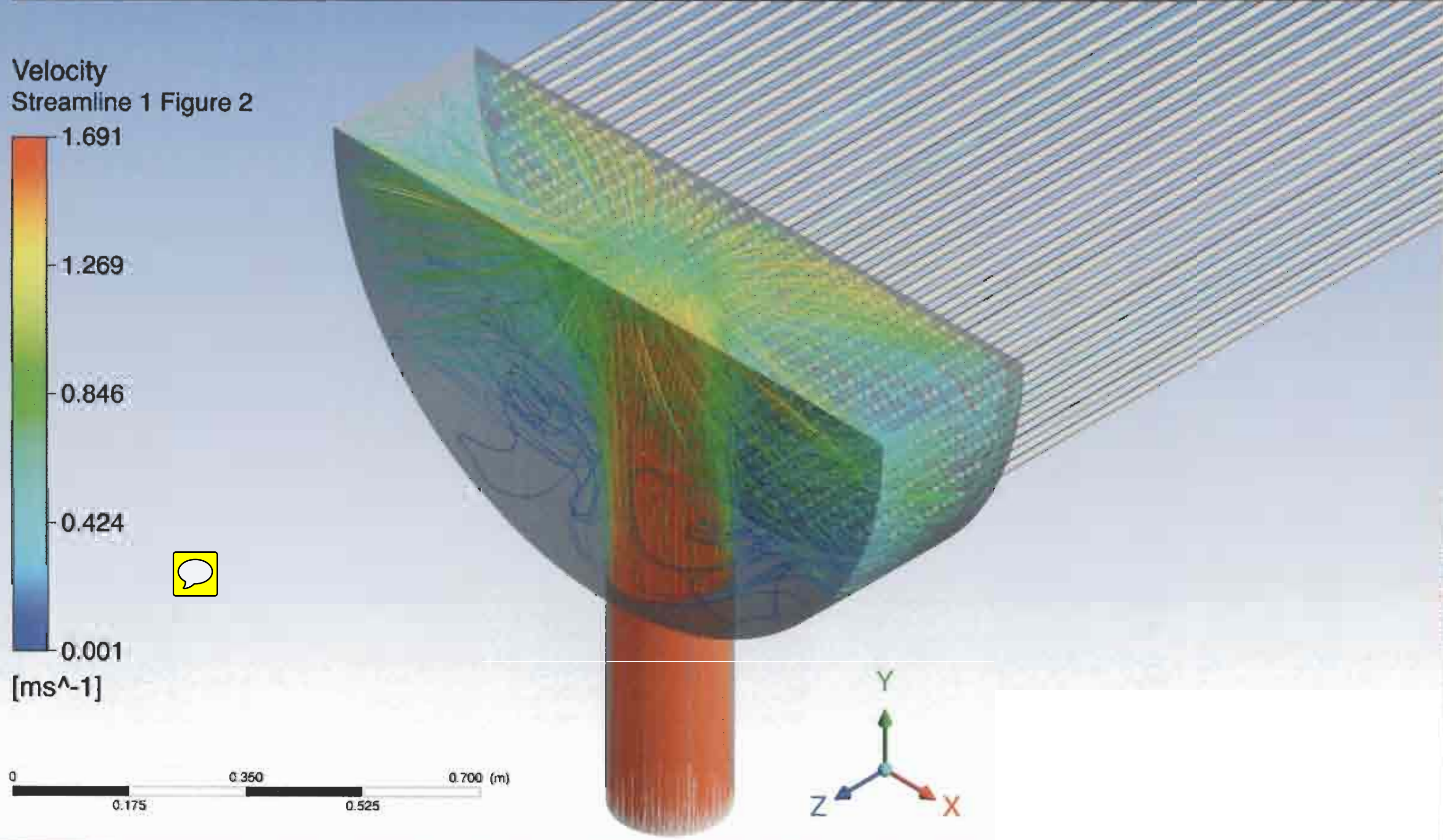
RESEARCH. EVALUATION. INNOVATION. SOLUTION.

CALGAVIN



SOLVING PROBLEMS FINDING SOLUTIONS

CFD SOLVES BOTH HEAT TRANSFER AND FLUID FLOW PROBLEMS IN HEAT EXCHANGERS

CALGAVIN has the capability and experience to model real-life heat transfer equipment in order to address mechanical, thermal and fouling effects on heat transfer, mass transfer and fluid distribution. Computational Fluid Dynamics (CFD) can be used as a powerful analytical tool allowing solutions to be developed to solve problems relating to;

- thermal deficiencies as a result of fluid maldistribution (e.g. optimising fluid distribution in exchanger headers)
- turbulent and laminar fluid flow with heat transfer
- selection of optimum flow velocity to meet minimum projected fouling characteristics in the design and simulation of heat exchangers
- heat sink optimisation
- radiant heat problems
- mixing simulations of multiple fluid streams
- single phase reactions
- erosion issues
- vibration problems

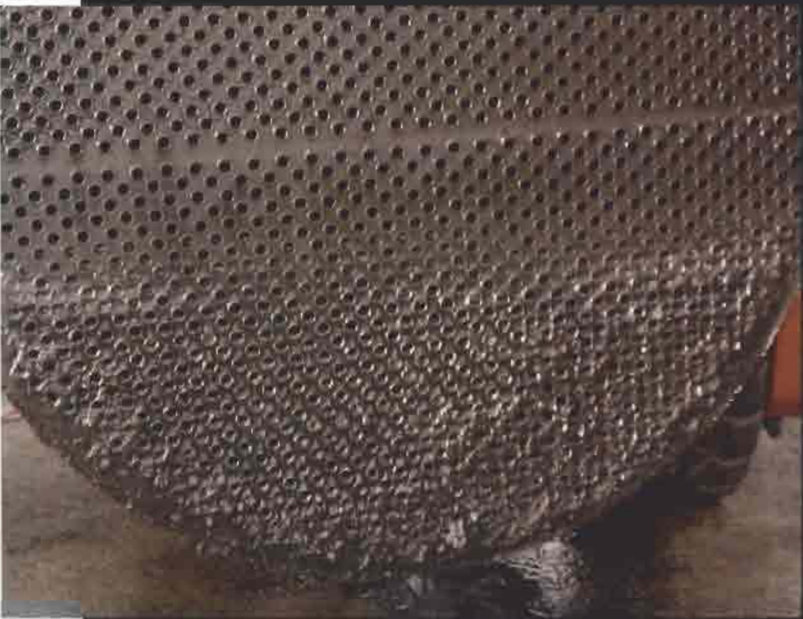


FIG. 1 A LAYER OF DEPOSIT SETTLES ON THE TUBE-SHEET

CFD BEING USED TO UNDERSTAND THE ONSET OF CRUDE OIL FOULING

CASE STUDY

A two pass AES TEMA type Shell and Tube heat exchanger heating VDU feed on the tube-side with the returning vacuum residue on the shell-side showed high fouling rates during operation. This was mainly caused by low fluid velocities ranging from 0.2m/sec to 0.5m/sec on the tube-side. Fig. 1 shows a layer of deposit settles on the tube-sheet after years of operation.

CFD was employed to investigate the flow pattern within the header and tubes. In order to simulate the isothermal flow behaviour, a CFD model with exchanger dimensions and actual plant data was created.

Fig. 2 The highest crude oil velocity in the header can be observed near the pass partition plate.

When simulating velocities at a plane adjacent to the tube entrance, a similar picture emerges. High crude oil velocities with resulting higher shear forces are present near the pass partition shown in Fig. 3. This explains the appearance of the tube-sheet after years of operation, the higher wall shear forces show a reduced risk of fouling in this area.

When setting the interrogation plane for lower operational velocities at half the tube length (Fig. 4) there is still a non-uniform fluid distribution within the bundle. Highest velocities near the top of the bundle (A) and lowest velocities observed in the middle of the bundle (B) show a difference of about 20%. These low velocity zones (B) have lower heat transfer coefficients and therefore higher wall temperatures can be expected. Owing to the lower wall shear forces in area B, there will be less potential for removal of the fouling layer.

This study shows that CFD can be used to explain how fluid distribution within a bundle can influence the onset of fouling.

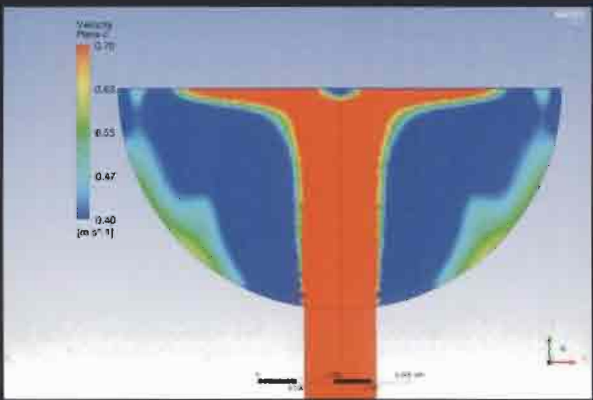


FIG. 2 VELOCITY DISTRIBUTION IN INLET NOZZLE AND HEADER

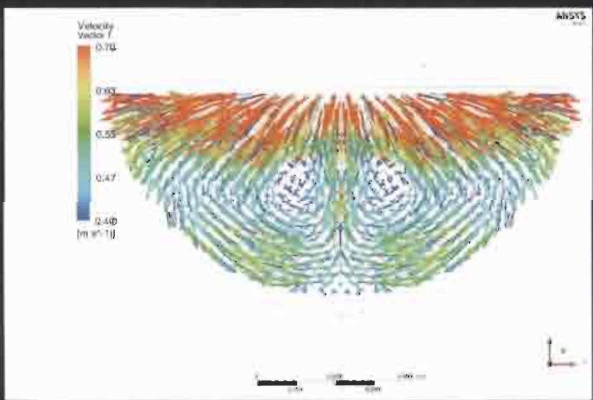


FIG. 3 VELOCITY VECTORS IN EXCHANGER INLET HEADER

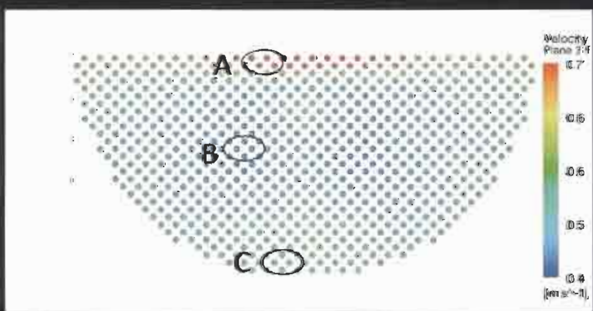


FIG. 4 VELOCITY VARIATION ACROSS THE BUNDLE PLANE

THE VALUE OF CFD IN IDENTIFYING CASES OF FLOW STRATIFICATION

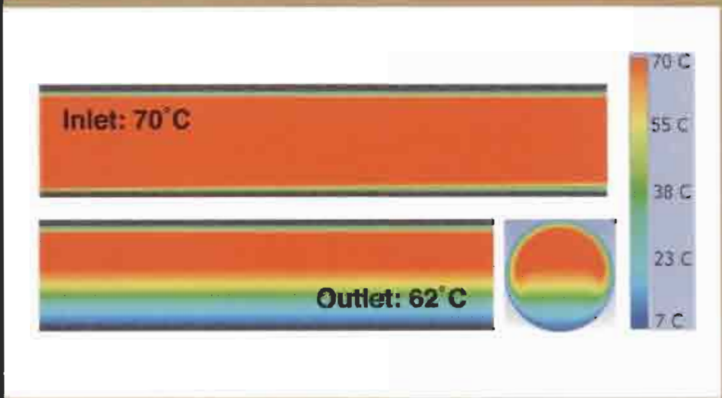


FIG. 5 CFD SIMULATION OF PLAIN EMPTY TUBE

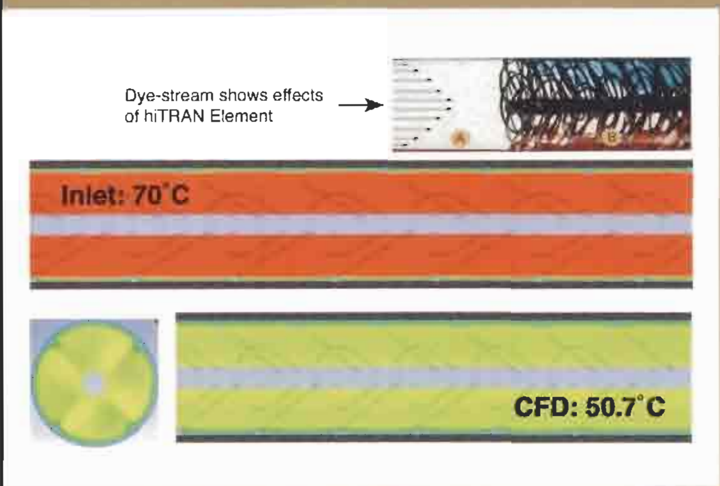


FIG. 6 SIMULATION OF TUBE ENHANCED WITH HITRAN MATRIX ELEMENT

CASE STUDY

This case study illustrates the even temperature distribution and velocity profile together with increased heat transfer in the presence of hiTRAN Matrix Elements compared to an empty tube. These elements induce shear and mixing effects which continuously remove the stagnant fluid from the wall and replaces it with fluid from the centre of the tube. As the tube-side heat transfer rate is increased the temperature of the wall and the tube-side fluid film will approach that of the bulk. Lowering the temperature difference between the wall and fluid film can reduce the rate of fouling when temperature-sensitive fluids are being heated or cooled. With increased film mixing particle deposition rates can be reduced, keeping tubes cleaner and the heat exchanger in service for longer.

Based on research data, Fig. 5 and 6 show comparative temperature profiles longitudinally through tubes of 2.5m test length. A plain empty tube and a tube with hiTRAN element are compared. (Reynolds number 253; mass flow 195kg/h; viscosity 12cP). Fig. 6 shows a much higher rate of cooling when enhancement is incorporated.

Empty tube characteristics (Fig. 5)

- Stratified flow
- Highest velocity at the top of the tube
- Long residence time at bottom of tube
- Low heat transfer

Enhanced tube characteristics (Fig. 6)

- Good fluid distribution
- Good temperature distribution
- High heat transfer with low outlet temperature
- Even velocity profile

FIG. 7 REFINERY AIR-COOLER BUNDLES

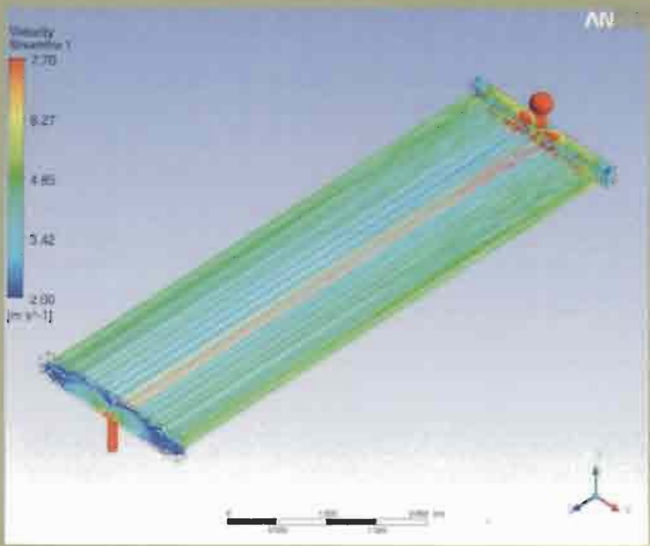


FIG. 8 AIR-COOLER BUNDLE SHOWING MALDISTRIBUTION

CFD SIMULATIONS TO IDENTIFY MALDISTRIBUTION IN AIR-COOLERS

CASE STUDY

This case study illustrates maldistribution in an air-cooled heat exchanger. This air-cooler has 134 tubes each 6m long with an inlet nozzle that enters vertically into the header. Simulations were carried out isothermally with gas as tube-side fluid. The maldistribution can be seen by the change in colour with the areas of low velocity (blue) and the areas of high velocity (red).

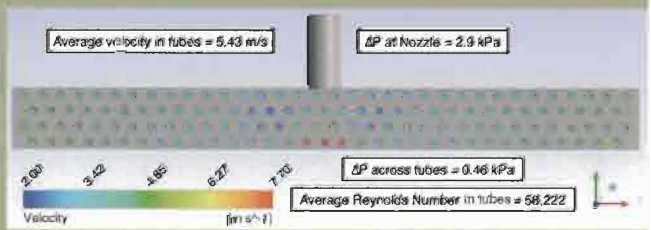


FIG. 9 AIR-COOLER HEADER SHOWING GAS VELOCITY VARIATION

TALK TO OUR ENGINEERS

CALGAVIN Engineers have first-hand knowledge and experience in the design and simulation of heat exchangers. We work with you to resolve heat exchanger and network problems and answer specific technical queries with regards to fluid flow and heat transfer.

CALGAVIN LIMITED MINERVA MILL, STATION ROAD, ALCESTER, WARWICKSHIRE B49 5ET, UK
TEL +44 (0) 1789 400401 FAX +44 (0) 1789 400411
EMAIL SOLUTIONS@CALGAVIN.COM
CALGAVIN.COM