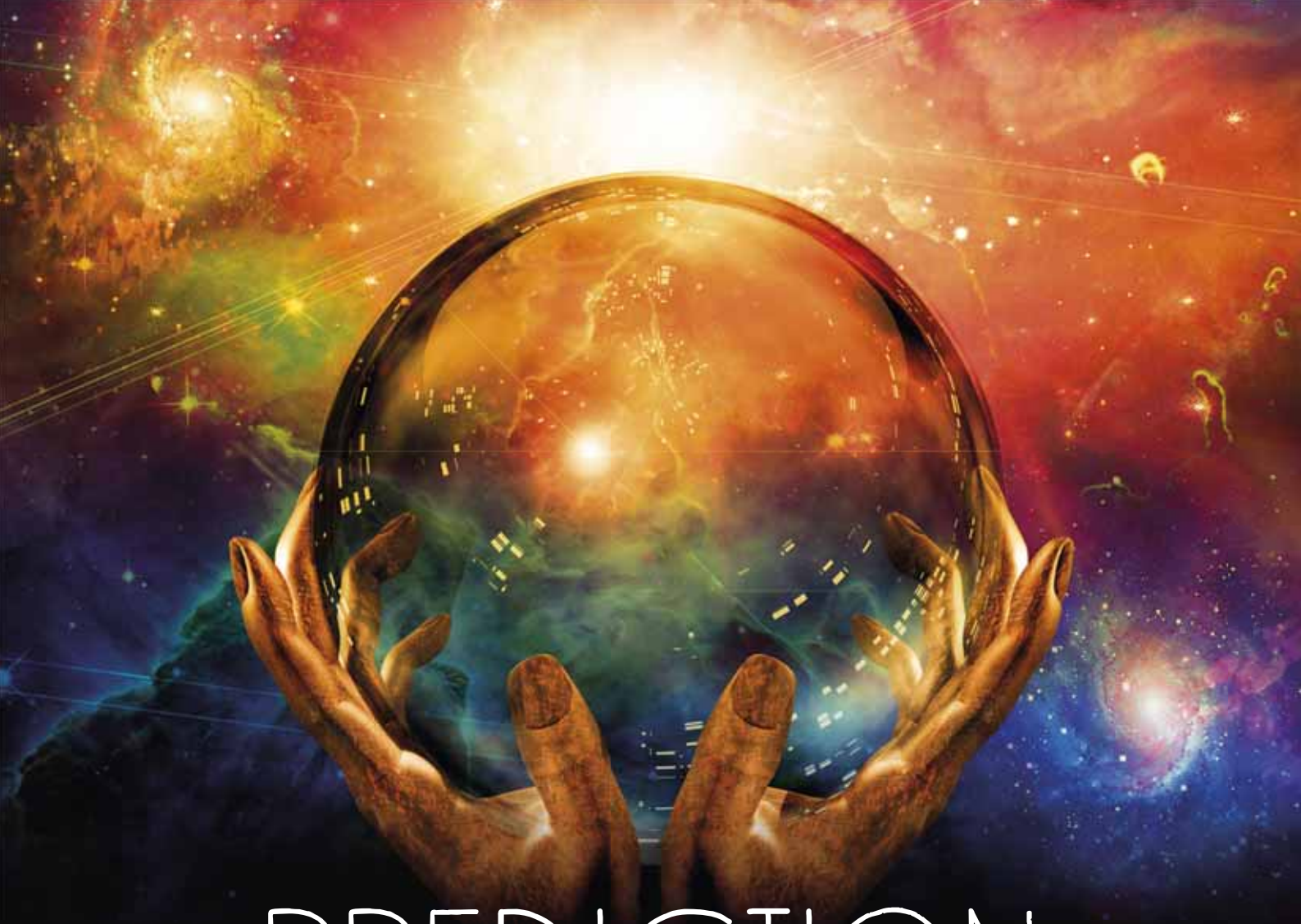


PREDICTION WITH PRECISION

Massimiliano Di Febo, Pasquale Paganini and Alfredo Costa, IPC, Italy, share CFD simulation tips for the process industry.

Computational fluid dynamic (CFD) simulations play an important role in machinery engineering and design processes. They are often used to complete the design loop that normally includes theoretical studies, previous manufacturer experiences and experimental tests.

Using CFD simulations derives from the need to reduce design, prototyping, and manufacturing times and costs. This method provides a way of getting first analyses and preliminary evaluations quickly, without the need to physically manufacture the system under study. Moreover, in many cases, CFD methods enable users to analyse systems under operative conditions that may be difficult to replicate on a shop floor. In special cases, CFD simulations can estimate some parameters that would be very difficult, or impossible, to measure directly (e.g. regions where it is not possible to access with transducers). These capabilities may be especially useful in cases where experimental models are either expensive, not available, or require long times.

In the oil and gas processing industry, there are many fields where CFD methods have proven to be important applications, from the design of gas turbine parts (blading, combustor, etc.) to entire stages of compressors or pumps, valves, auxiliary machinery, and plant devices.

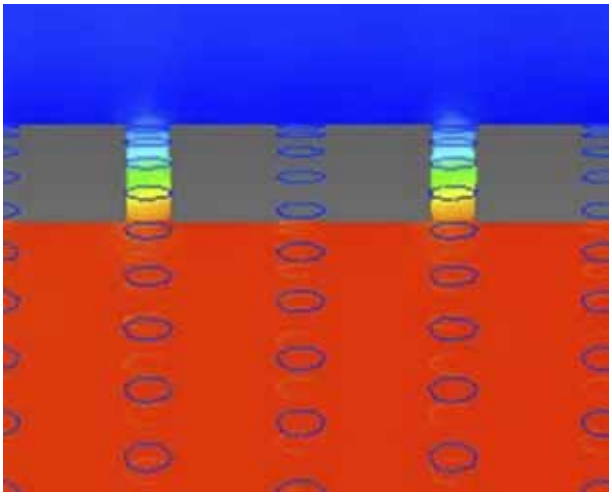


Figure 1. The pressure in the hole.

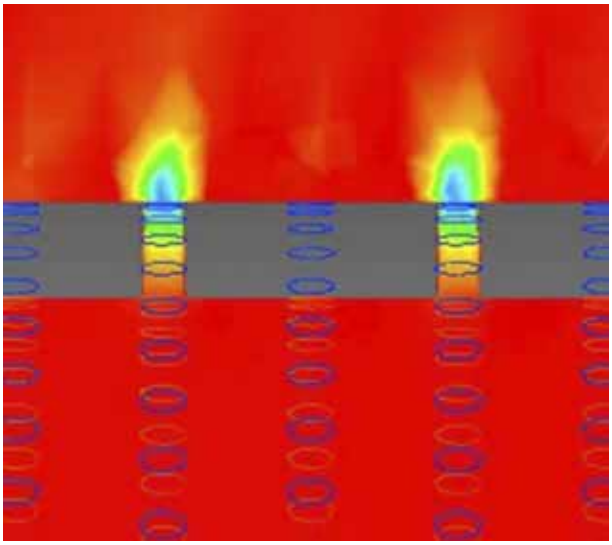


Figure 2. The temperature in the hole.

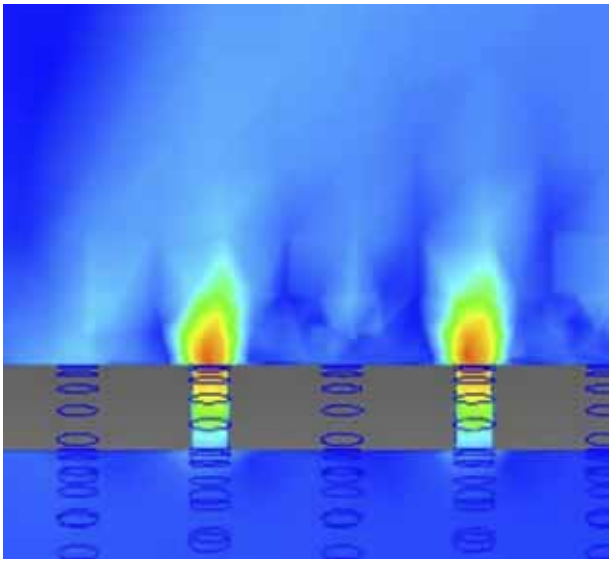


Figure 3. The velocity in the hole.

Case study

This article will present and describe the main steps of a generic CFD simulation. An example is considered to better highlight the way in which the simulation has been carried out. In the specific case that is considered, the CFD simulation has been developed for an auxiliary pipe device.

Beyond the specific function of the device, this article aims to provide a practical description of how the simulation has been developed. In fact, while there are many powerful simulation software packages available, offering a large number of features, it remains the responsibility of the CFD specialist to determine the settings that will be necessary or beneficial for the final quality of simulation results, according to each specific application peculiarity. For these reasons, experiences and methods should be shared among analysts in order to contribute to the diffusion of the CFD methodology for the overall benefit of the industry.

Based on these considerations, this article is organised to give a summary of the simulation process, sharing key considerations about topics that each analyst should approach when executing a typical simulation in the industry process environment.

The plant device considered in this article was installed just downstream of a control valve to reduce the temperature, pressure and noise while smoothly redirecting the gas flow from high pressure environments to low pressure ones. The CFD simulation was requested to check the design data defined in a previous design stage on a theoretical basis, with an intention of comparing the obtained result with experimental measurement.

It is important to recall that any CFD simulation should be understood as part of an overall product development cycle that includes experimental testing, and that may need to be reiterated in order to spread the maximum benefits. Standalone simulations may also be considered but preferably when resulting from an already completed simulation loop, for objects with parametric characteristics, and after accurate sensitivity analyses. Different simulations may represent a risk in the measure in which the difference of calculated parameter from real ones is unknown.

This article will describe the fluid dynamic simulation activity for the compressible flow inside the considered device. The obtained CFD results will then be compared to the available experimental test data and to the parameters that were predicted according to theoretical evaluations. The main simulation goals and operative steps will be outlined, along with preferential software settings and relative reasons for their use.

Main goals

The main goals of the CFD for the system in this case study are as follows:

- Verify the customers' specifications and validate the resolution model through experimental testing.
- Check for any critical flow conditions with particular reference to the possibility of condensation and cavitation.
- Get a picture of the flow field inside the device in order to be able to predict the main stationary flow phenomena.
- Provide any indications and precautions to optimise the device against erosion phenomena, mechanical stress, noise and vibration, as well as to support detail and construction engineering.

The simulation activity was arranged in the following three phases.

Phase one – pre-processing

In this case, the device consists of two main environments (inlet and outlet) while the passage area is represented by 0.25 in. holes spaced along the cylindrical surface. In phase one, the calculation domain for the simulation is generated via computer-aided design (CAD) and, consequently, the computing mesh is created over the wet volume.

After defining the computing mesh, the physical properties and boundary conditions are assigned to flow domain. The operating fluid considered in the simulation is a hydrocarbon modelled as real gas. The inlet and outlet conditions for the device were available from customer specifications in terms of pressure, temperature, density, viscosity and mass flow.

The walls are considered to be a smooth metal surface and the simulation is adiabatic (without heat transfer effects).

Particular attention has been used in refining the mesh in regions where the flow was expected to manifest particular characteristics, particularly in proximity of the radial holes.

The mesh density has been locally increased in areas of major interest about particular flow phenomena, and adequately sized to allow the observation of turbulent phenomena. For example, inside the holes, the mesh size has been reduced to one-fifth of the size applied in remaining regions. Consequently, the mesh density changes along the control volume and it appears to be linked to the specific geometry scale of the device in consideration.

The device in consideration is static in nature and the flow has been considered as stationary.

For turbulence, the 'k-epsilon' model has been used because it is typically more accurate than the 'constant eddy viscosity' models, even if it is more computationally intensive. It is a general purpose turbulence model that performs well across a large number of applications.

Phase two – solving

The simulation raced for 2500 iterations. Throughout the solution process, the CFD software solves the equations that govern the physics of the flow in the defined model domain. During this stage, the average values of computed quantities must be monitored until there is satisfactory convergence. The primary criteria for determining the convergence is that the change of each degree of freedom should be minimised over a wide range of iterations. In this simulation, the stabilisation of solutions within 1% was considered as 'good convergence'.

Phase three – post-processing and analysis

In phase three, the results of the simulation have been visualised and subsequently analysed. The flow fields generated by the simulation were studied, including visualisations of pressure, temperature and velocity fields and flow field streamlines. Such visualisation allowed an understanding of flow behaviour within the device under consideration.

The holes passage represents the region where fluid dynamics phenomena are the most significant and requires a more in-depth investigation.

Figures 1 – 3 show a detail of the area for a better understanding of the distribution of pressure, temperature and speed, respectively, in this particular region.

Table 1. Deviation between CFD results and field test

	Percentage error (%)
Inlet pressure	0.5
Outlet pressure	0.2
Inlet temperature	0.6
Outlet temperature	1
Mass flow	0.8

The results of the simulation show how the gas flow undergoes a strong expansion (almost all the pressure drop between the inlet section and the outlet section) in the hole section and, simultaneously, the gas temperature decreases very quickly. This phenomenon is consistent with the expected behaviour due to large volumes of the inlet and outlet ducts.

There is a strong flow acceleration in the velocity section holes that is up to 10 times greater than the flow upstream of the passage hole.

In this region, the flow undergoes a sudden acceleration (Mach number greater than 1) and then faces a progressive pressure recovery up to the discharge section.

Numerical results

Table 1 show the percentage error between the CFD results and field test. For the inlet and outlet temperature and mass flow, the CFD values are in good alignment with the field test measurements. In general, the simulation indicates flows that are slightly higher than experimentally measured values.

Interestingly, the simulation in this case highlighted that, locally, the flow inside the device reaches a supersonic regime in the holes' passages (minimum section), while the Mach number remains sub-sonic in the inlet and outlet environments. This result helped to confirm the need to take adequate design solutions that aim to allow the component under consideration to resist the acoustic and structural stresses caused by the intense acceleration of the internal flow. Therefore, this specific simulation experience confirmed the CFD's capability to contribute to the overall product development and improvement loop.

It should be noted that for a valid comparison of the field measurement, it is necessary to have a good correspondence between the location where transducers are physically placed and the model regions where the boundary conditions are imposed.

Final considerations

CFDs continue to help model real world problems effectively and offer insight into complex flows. The adoption of CAD practices allows for an in-depth investigation and understanding of the actual stress distribution and flow patterns inside plant devices and machinery, enabling their coupled effect (flow-structure interaction) to be predicted.

Looking to the future, it is reasonable to expect that continuously increasing computational power and the availability of reliable refined codes will allow for consistent improvements to process machinery and the design processes of devices. It should also be recalled that with desirable augmented software capabilities, CFD is not meant to replace designer and original equipment manufacturer experiences. The overall design loop needs to be closed with a final device for physical testing.



HYDROGEN COMPRESSION FOR MULTIPLE NEEDS

THE NEA 360° PORTFOLIO TO BOOST HYDROCRACKERS

Hydrocrackers comprise the heart of the refinery. Inside the giant reactors, heavy waxy feedstock is converted into lighter products such as jet fuel, kerosene and diesel fuel. Today's crackers need large quantities of hydrogen to convert high-sulfur material into low-sulfur fuels for vehicles and airplanes under high temperatures of up to 800°C/1,500°F.

As hydrocrackers require major invests, reliable equipment such as reciprocating compressor packages are mandatory. The **NEA heavy-duty hydrogen compressor** is named as the best-in-class solution for hydrocrackers due to its tailor-made design and flexibility in driving high volume flows of up to 100,000 Nm³/h at some hundred bar discharge pressure and up to 30 MW power. As reliability counts, no single installation is left unattended. From start-up to maintenance and professional diagnostics, the service provider **NEAC Compressor Service** keeps the compressor running. Worldwide, with full life-time support.

Authorized OEM supplier for reciprocating compressor lines:

NEA · WÜRZEN · Eslingen · Chicago Pneumatic · KSB COMPRESSORS · ES · Linde · HALBERG · Demag · Hofer · Borsig (recips built in Berlin up to the end of 1995) · PPC PENN Process Compressors