Mechartés

SIMULATION EXPERTS

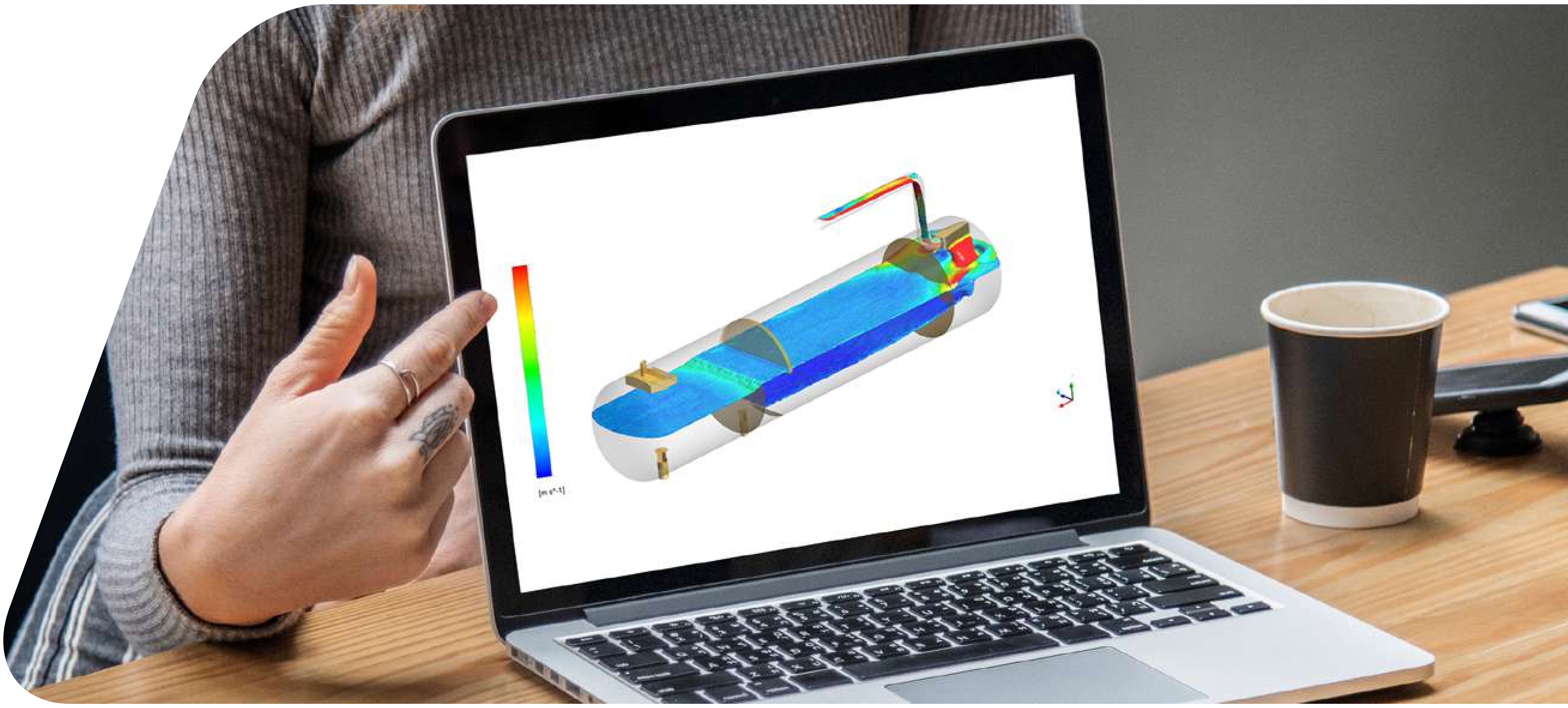
Multiphase CFD Simulation for Separators Pressure Vessel

Table of Contents

Introduction

ASME Section VIII Division 1, is the world's most widely used Pressure Vessel Design Code. Mechartes offers complete analysis services from ASME Section VIII, Division 1 Rules for the Construction of Pressure Vessels, to Division 2 Alternative Rules. Whether the task involves designing new equipment, analyzing existing equipment, optimizing design, or interpreting code requirements and options, our Engineers will ensure your company is fully equipped to tackle its unique challenges.

CFD is widely used to calculate pressure drops, heat transfer rates, design massflow rates, and fluid dynamic forces such as lift and drag. Hence, CFD is becoming an increasingly important engineering design and a research tool. Due to the advances in numerical solution methods and computer technology, geometrically complex cases can be treated like those often encountered in turbomachinery.



Introduction

CFD complements the approaches of pure theory and pure experiment in analyzing and solving fluid dynamic problems. The CFD simulation explains the microscopic and macroscopic features of the single-phase, two-phase, or three-phase separation phenomenon.

Mechartes has complete expertise in the following codes for separators and pressure vessel design based on the following Codes and Standards:

- ASME BPVC Section VIII, Division 1, Rules for the Construction of Pressure Vessels
- ASME BPVC Section VIII, Division 2, Alternative Rules
- ASME BPVC Section VIII, Division 3, Alternative Rules for Construction of High-Pressure Vessels
- ASME Pressure Vessels for Human Occupancy (PVHO)
- ASCE 4-98 and ASCE 7-02
- AISI N690
- ABS Underwater Vehicles, Systems & Hyperbaric Facilities
- API 579 / ASME FFS-1, Fitness for Service

How CFD Modeling of Multiphase Flows Is Helping the Oil and Gas Industry

How CFD Modeling of Multiphase Flows Is Helping the Oil and Gas Industry

CFD is best used where the system behavior cannot be calculated using conventional methods and needs complex mathematical modeling to capture the behavior. This technique allows designers to verify that their products will comply with a customer's specifications early in the design cycle, accelerating the product development process.

Transient Multiphase CFD (Computational Fluid Dynamics) analysis is a numerical simulation technique used to study the behavior of two or more immiscible fluids (such as oil, gas, and water) in a pressure vessel, such as a separator. The analysis involves solving the equations of fluid dynamics and species transport to predict the flow patterns, pressure drop, and phase distribution in the vessel. This information can be used to optimize the design of the separator for improved performance and safety.

Solid particles often occur in most industrial processes involving fluid flows. Typically, in the oil and gas industry, the sand particle is frequently produced and entrained along with crude oil and natural gas through pipeline transportation before reaching any process equipment.

How CFD Modeling of Multiphase Flows Is Helping the Oil and Gas Industry

During the multiphase flow (presence of more than one phase), particles obtain momentum from the carrier fluid and flow along to intrude on the inner wall of the pipes, fittings, valves, and other pumping devices by causing wear to these facilities. CFD experts can perform multiphase flow analysis of flow within pipelines and various equipment to analyze the effect of the different states of fluid on the life of the equipment. Based on the analysis, various effects such as solid erosion damages are estimated, and consulting is provided by experienced CFD consultants like Mechartes, to minimize the damage due to the erosion in this equipment.

We can perform various multiphase CFD flow analyses

- To optimize the equipment design and minimize failure which may lead to potential oil spillage
- To validate the design operating conditions and if the desired outputs are not achieved, than modifications are suggested based on CFD results
- Production loss, resulting in significant financial losses as well as safety and environmental concerns
- Design flow separator devices that segregate the solid and the fluid particles.

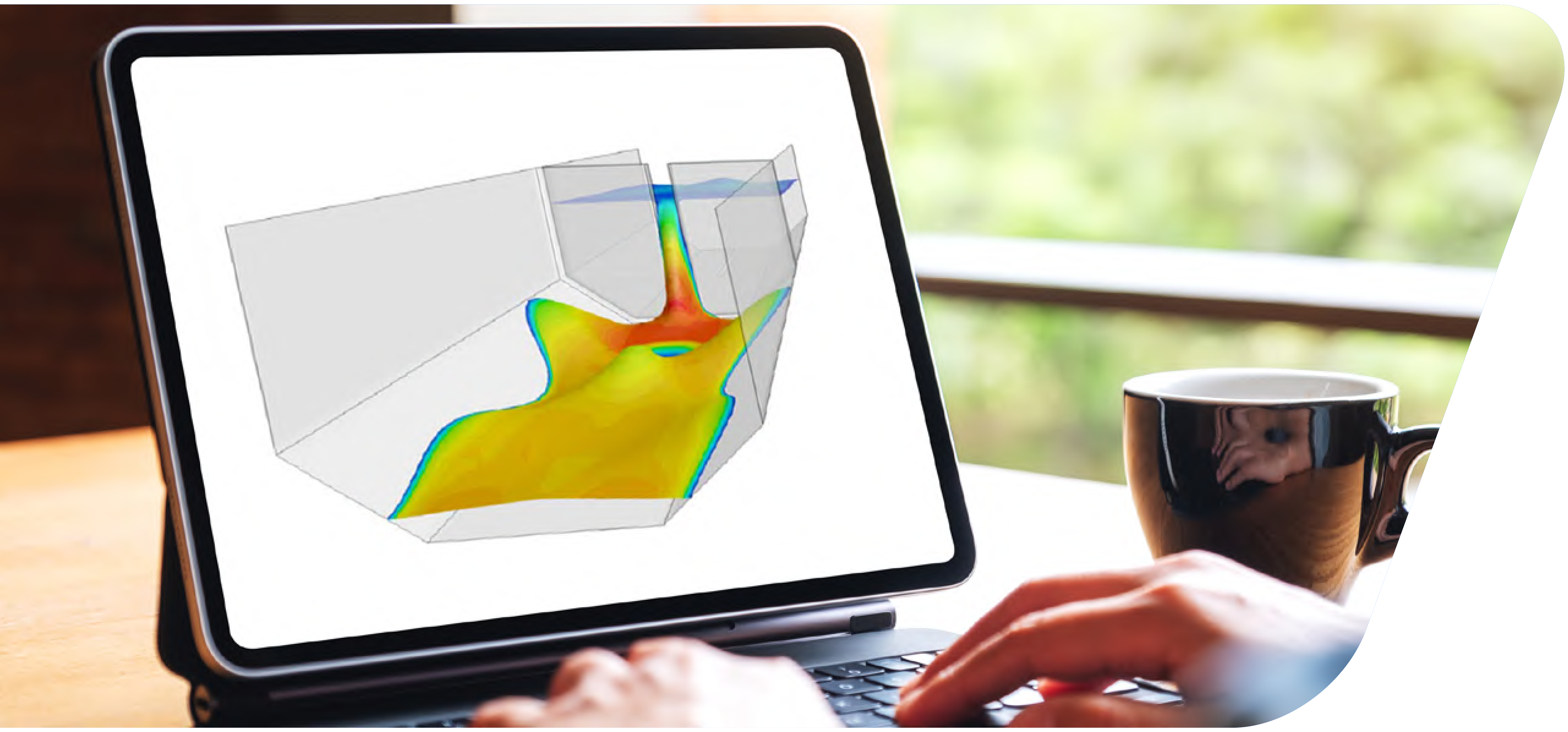
Figure-1

For Separator vessel, transient and multiphase CFD analysis can be used to determine the oil in water, at water outlet and water in oil, at oil outlet, and oil and water at gas outlet, and also to calculate the forces acting on the internal parts of the vessels. Based on the results obtained, CFD experts like Mechartes can suggest the design modifications.

How CFD Modeling of Multiphase Flows Is Helping the Oil and Gas Industry

Multiphase CFD analysis is also used in predicting the three-phase gas-liquid-solids or two-phase gas-liquid and liquid-liquid flow behavior to provide continuous and safe operations or transportation of production fluids. These flow simulation services provide instrumental data to the clients, such as flow patterns, liquid holdup, and pressure drop in complex flow conditions, thus optimizing the design of the equipment or pipelines.

Multiphase CFD analyses are turning out to be a boon, especially in the oil and gas industry, where there is a need to maximize production and minimize any delays or losses. These analyses not only help prolong the life of the equipment required in oil and gas but also ensure that there are no lethal environmental disasters due to the same.



Multiphase Separator Terminology

Multiphase Separator Terminology

Multiphase separation can be carried out through various oil processing equipment with the specific terminology corresponding to each system. Hence, it is worth defining the most important multiphase separators in the oil industry before proceeding:

- The conventional oil and gas separator, which is normally installed on a production facility or platform, may be referred to as an "oil and gas separator", "separator", "stage separator", "production separators", "test separators", etc.
- A "knockout vessel" removes water or all liquid from the well fluid flow.
- An "expansion vessel" is the first stage separator vessel usually operated at a low temperature.
- A "flash chamber" or "flash vessel" typically refers to a conventional oil and gas separator operated at low pressure as the second or third stage of the multistage separation.
- A "gas scrubber" is an oil and gas separator with a high gas to liquid ratio.
- In a "wet-type gas scrubber", dust, rust, and other impurities of the gas phase are washed using a bath of oil or other liquid, and the gas flows through a demister to further remove liquid droplets from the gas stream.
- A "dry-type gas scrubber" or "gas filter" is equipped with demisters and other coalescing media to aid in the removal of most of the liquid from a gas stream.

Separator Internals

Internals such as inlet diverters, controls, flow-distributing baffles, and mist extractors enhance the separator efficiency. A “weir” design horizontal three-phase separator and a vertical three-phase separator, both with the installed internals, are shown in Figures 1.1 1.2

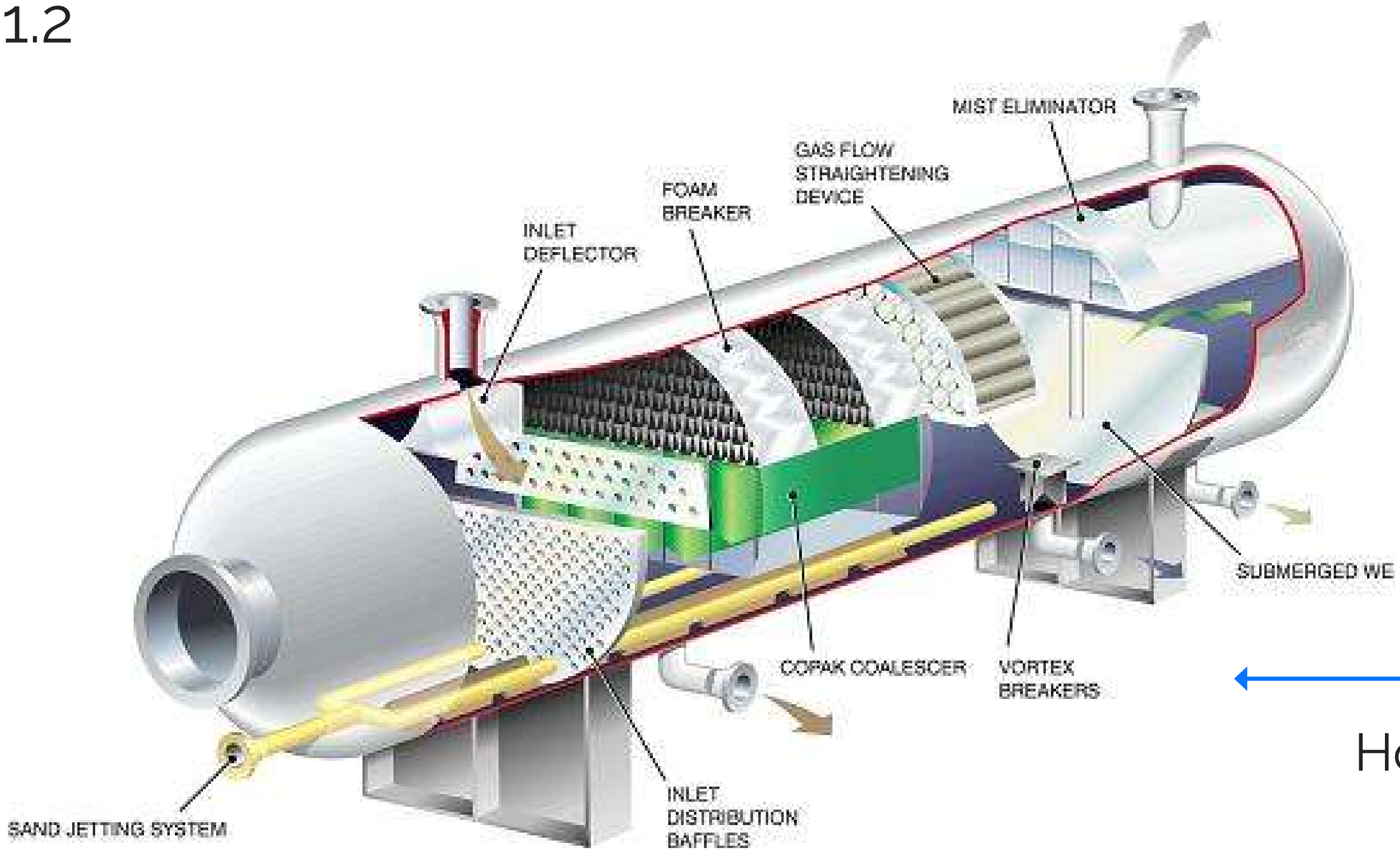
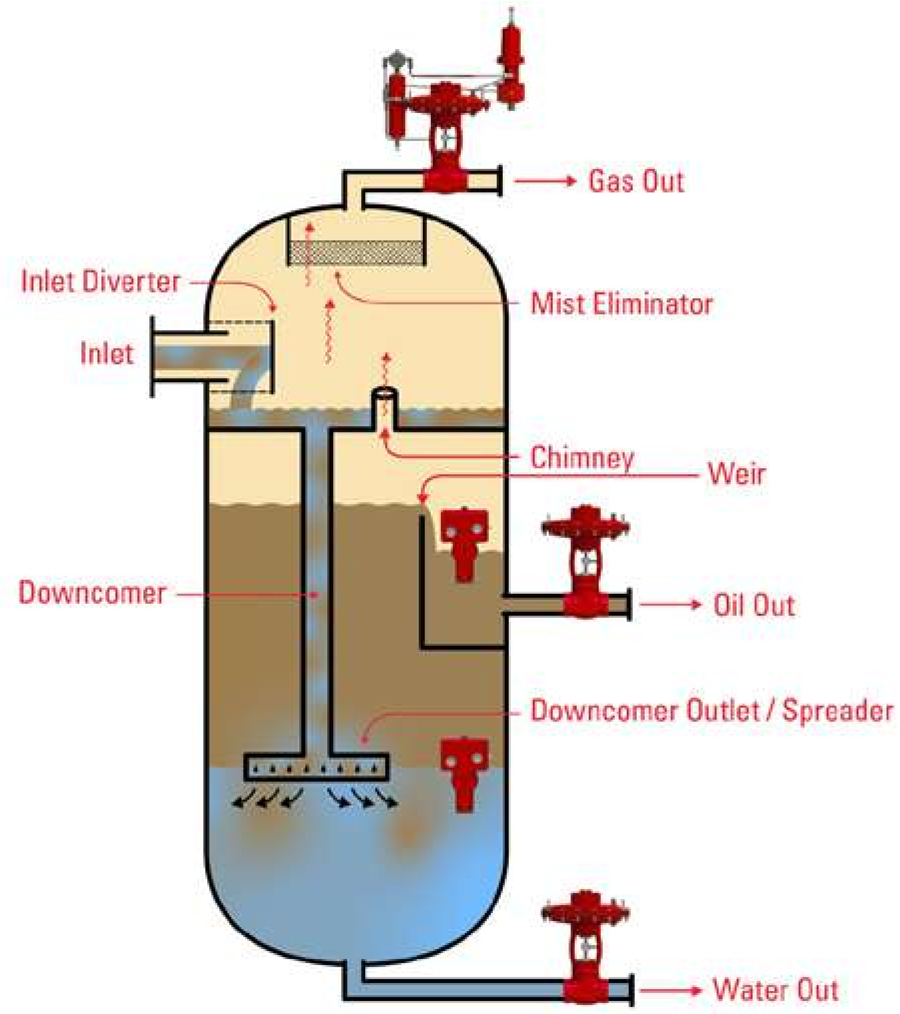


Figure 1.1:
Horizontal Separator

Figure 1.2:
Vertical Separator



Multiphase Separator Analysis

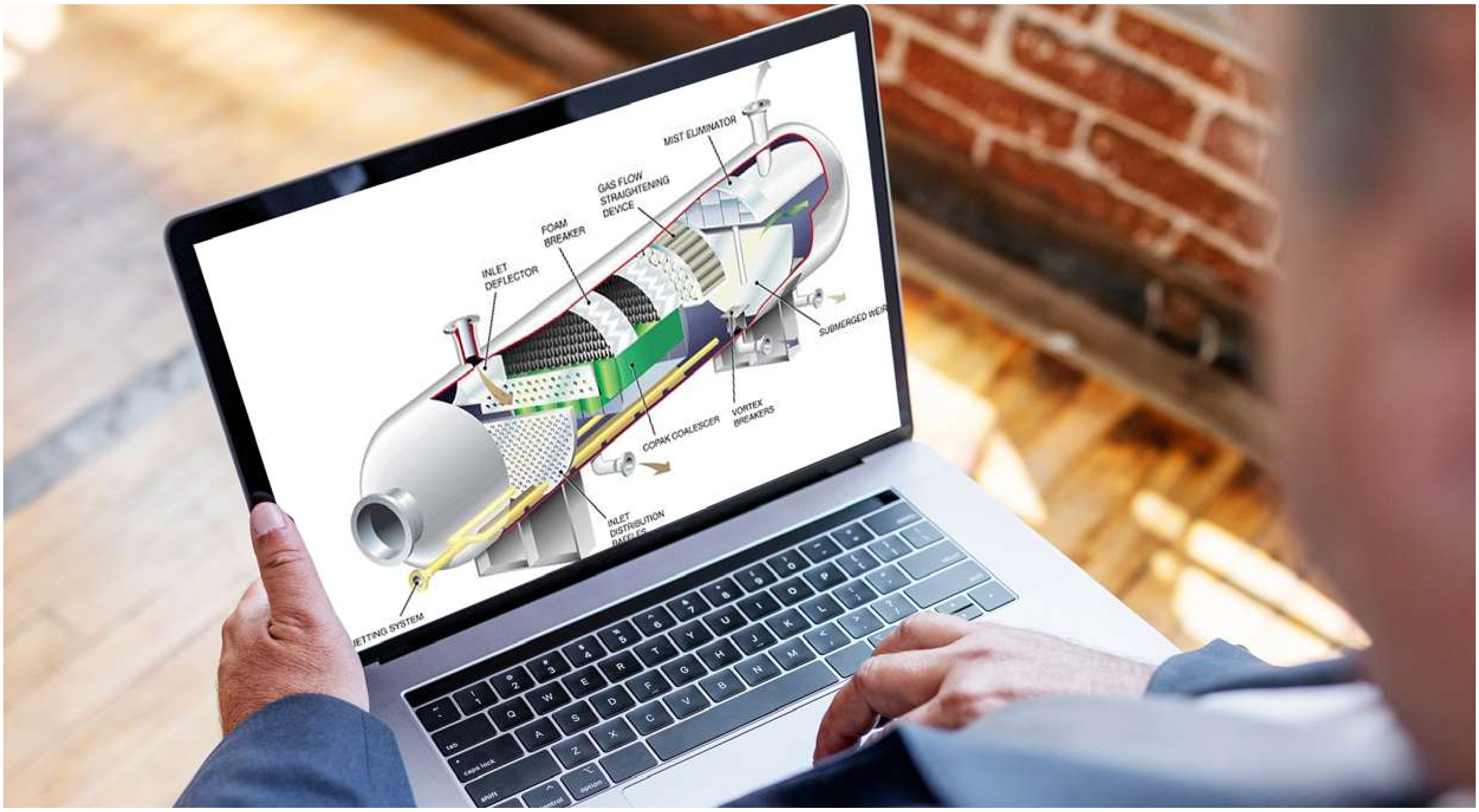
The multiphase separators are generally the first and largest process equipment in an oil production platform. Furthermore, this primary separation step is a key element in the oil and gas production facilities in that downstream equipment, such as compressors, completely depend on the efficient performance of these multiphase separators.

“A fluid is anybody whose parts yield to any force impressed on it, and by yielding, are easily moved among themselves.” The physical features of any fluid flow are governed by three fundamental physical principles:

Multiphase Separator Analysis

- Mass is conserved
- Newton’s second law applies
- Energy is conserved

The results of the transient / multiphase CFD analysis can be visualized using graphical tools, such as contour plots and streamlines, to better understand the flow patterns and phase distribution in the separator. The results can also be used to optimize the design of the separator for improved performance and safety.



Strategies and Types of Model for Simulation of Multiphase Fluid Flows

In a multiphase CFD analysis for separators, the following factors are considered:

Geometry:

The geometry of the separator, including its dimensions, inlets, outlets, and internal structures (such as baffles or coalescers), is modeled in the simulation.

Fluid properties:

The physical and chemical properties of the fluid phases, such as density, viscosity, surface tension, and inter-phase transfer coefficients, are input into the simulation.

Boundary conditions:

The inlet and outlet conditions, such as flow rates, pressures, and temperatures, are specified in the simulation.

Turbulence model:

A turbulence model is used to predict the effects of turbulence on the flow and phase distribution in the separator.

Solver:

A numerical solver is used to solve the equations of fluid dynamics and species transport and predict the flow patterns and phase distribution in the separator.

There are two approaches to modeling multiphase flows:

- The Euler-Lagrange approach
- The Euler-Euler approach

In the Euler-Lagrange approach, a continuous fluid phase is modeled by solving the time-averaged Navier-Stokes equations. The dispersed phase is simulated by tracking a large number of particles through the flow field based on Newton's second law.

The Euler-Euler approach, however, deals with the multiple phases as continuous phases that interact with each other. Since the volume of a phase cannot be occupied by the other phases, phase volume fractions are assumed to be continuous functions of space and time, and their sum is equal to one.

i. Discrete Phase Model (DPM)

Following the Euler-Lagrange approach leads to the Discrete Phase Model (DPM) in Fluent. This model works well for flow regimes where the discrete phase is a reasonably low volume fraction, usually less than 12% (Fluent 6.3 User's Guide, 2006).

Various forces are taken into account while Fluent tracks the particles through the flow field, including the gravity force, the drag force (with the option of involving dynamic drag coefficient to account for particle deformation), the virtual mass force (accelerating the fluid surrounding the particle), the thermophoretic force (exerted on small particles suspended in a gas phase with a temperature gradient), the Brownian force, and the lift force.

DPM can also model the coalescence of particles and their breakups. For this purpose, based on the particle Webber number, a proper model within the spray model theory is used.

ii. VOF Model

The VOF model is a surface tracking model designed to simulate immiscible multiphase flows where the position of the interface between any two adjacent different phases is of interest. In the VOF model, the fluids share a single set of momentum equations, and the volume fraction of each fluid in each computational cell is tracked throughout the domain.

Applications of the VOF model include:

- Free-surface flows
- Sloshing
- The motion of large bubbles in a liquid
- The motion of liquid after a dam break
- The simulation of a jet breakup
- The steady or dynamic tracking of any liquid-gas interface

iii. Mixture Model

In the Mixture model, the phases are assumed to be completely interpenetrating. The Mixture model solves for the mixture momentum equation and the dispersed phases are modeled via calculation of their relative velocities.

Applications of the mixture model include bubbly flows, sedimentation, and cyclone separators.

iv. Eulerian Model

The Eulerian model solves a set of momentum and continuity equations for each phase. The pressure and inter-phase exchange coefficients are defined among phases based on the type of phases involved.

Applications of the Eulerian multiphase model include bubble columns, risers, and fluidized beds.

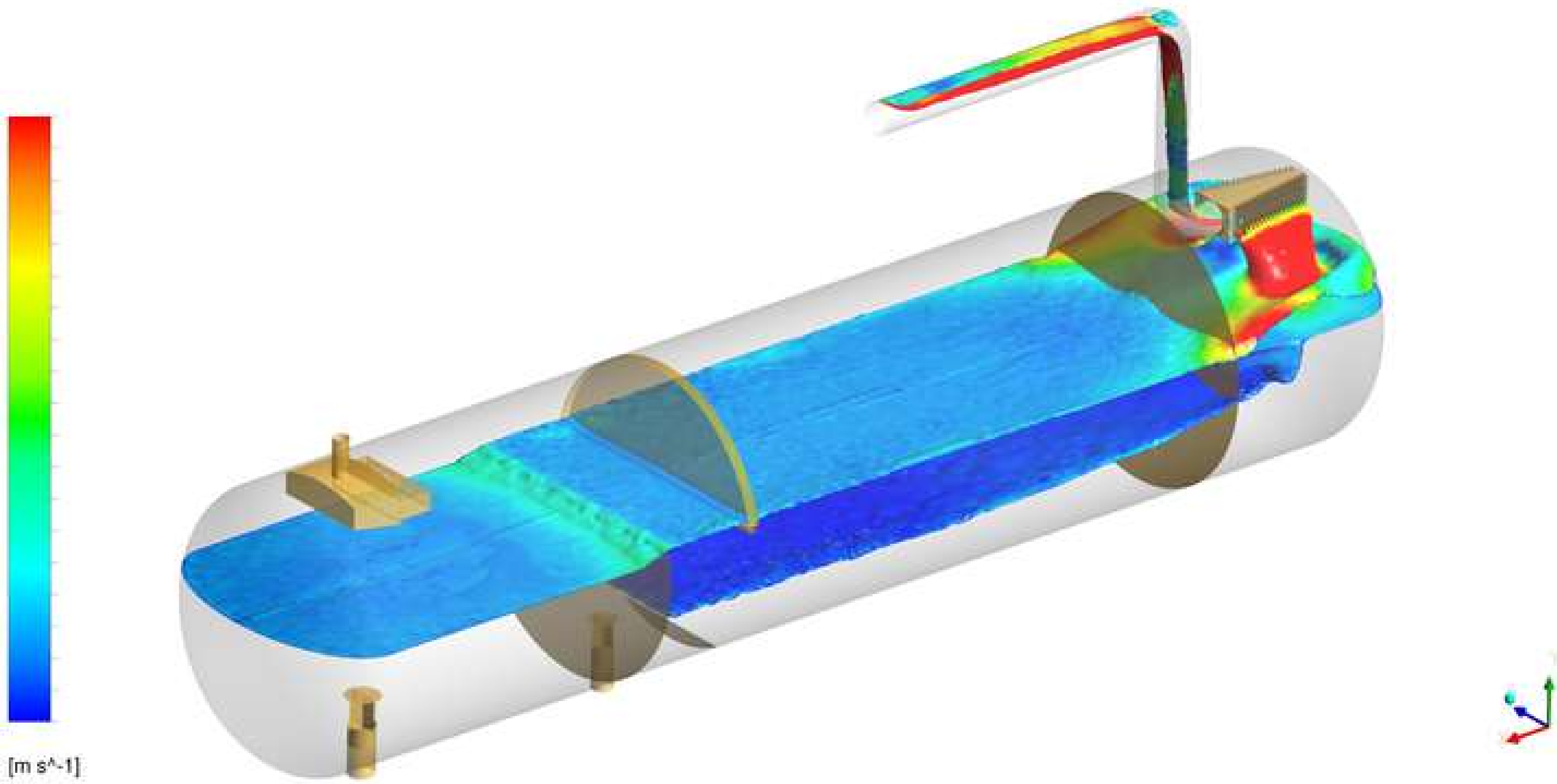


Figure 3

Why CFD should be part of your separator design?

Why CFD should be part of your separator design?

In today's market, cost reduction is front and center for all. CFD analysis helps in all project phases, from reducing initial project costs to operating costs.

Using CFD, the flow patterns in upstream piping, inlet devices, and within the separator can be predicted to ensure adequate residence time. Each separation mechanism can be studied, such as estimating the tendency for liquid droplet re-entrainment due to high gas velocities and improving the performance of internals by increasing the uniformity of the flow to the demisters.

Setting up a CFD study involves the following steps

Define the domain of interest:

What is the geometry, where does the flow enter and exit, and what are the boundary conditions, such as the velocity at the inlet etc.?

Mechartes has full capability to extract the CFD model from given 2D/3D CAD drawings and identify the boundary conditions from the given inputs.

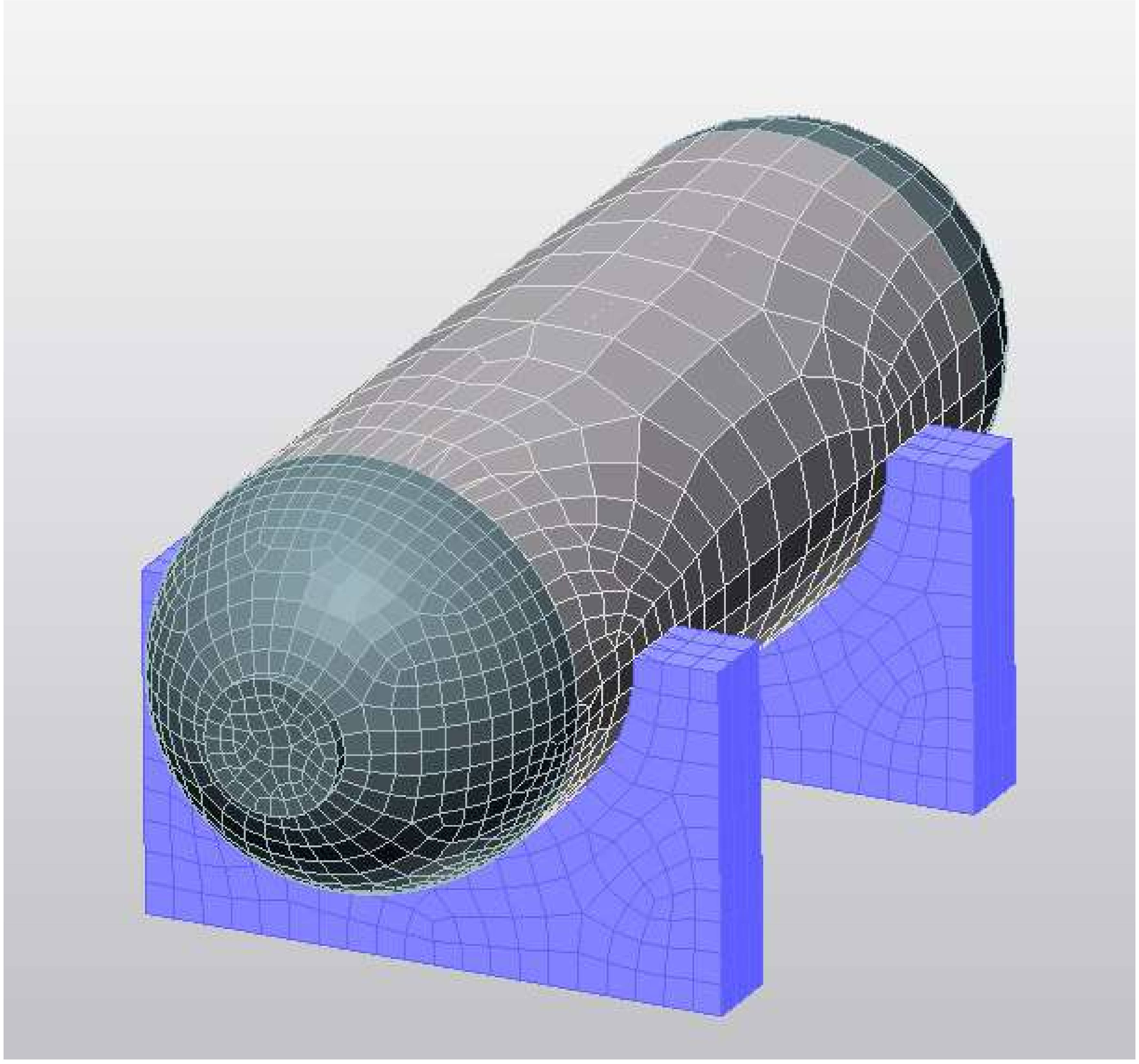
Discretizing or meshing the domain:

Rather than solving for the fluid behavior (velocity, pressure, etc.) at every point in space, we segment the volume using a mesh and then solve the equations at the center of each cell in the mesh.

Why CFD should be part of your separator design?

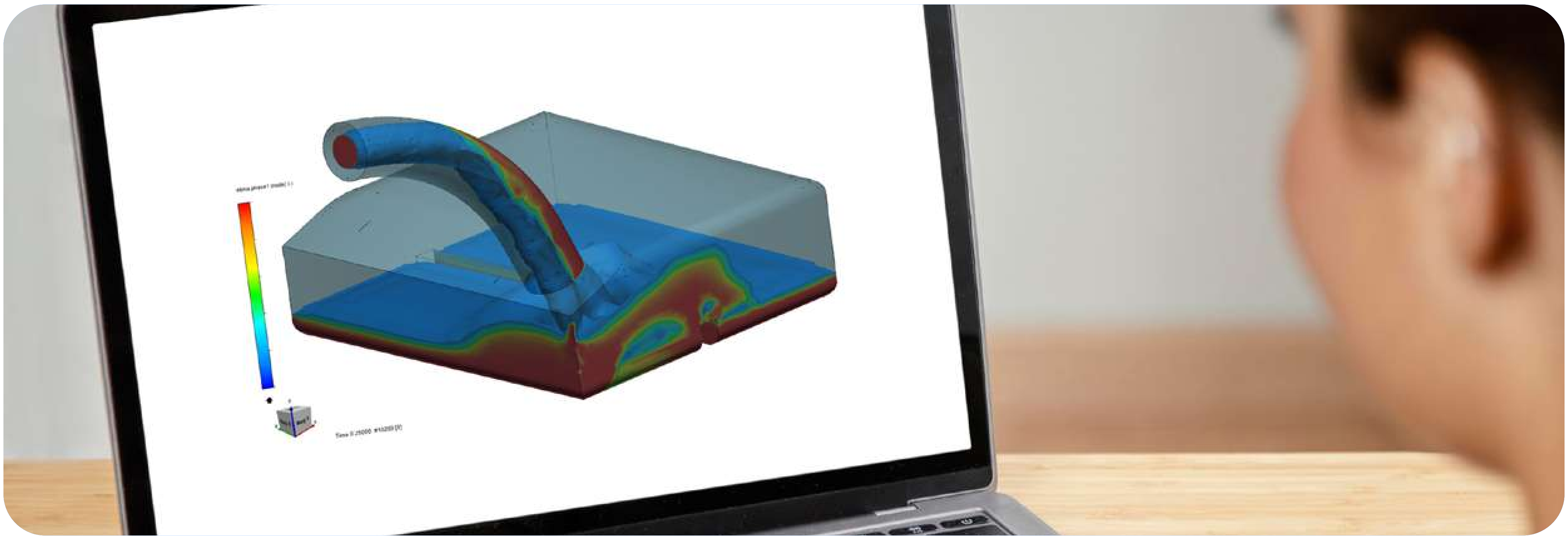
Think of a separator filled with Lego bricks; CFD will tell you the velocity, pressure, temperature, etc., at the center of each brick. For completeness, in time-varying problems, time is discretized by solving for time increments, such as every 0.1 seconds.

Secondly, the domain (geometry) needs to be discretized or meshed. This is an important and potentially time-consuming part of the analysis, and a good mesh begets good CFD.



Why CFD should be part of your separator design?

The mesh (number, size, and type of cells used) can influence the answer. For example, if a CFD simulation tries to simulate a vortex using one cell, the solver only has one point at which it calculates the velocity and pressure to represent the vortex. As the number of cells increases, decreasing the size of each cell, the resolution and, therefore, accuracy of the representation of the vortex improves. A similar approach is used to ensure or minimize the influence of the mesh over the solution; the mesh is progressively refined (reducing cell size) until quantities of interest, such as pressure drop, stop changing.



Select which physics to simulate: is the flow single or multiphase, thermal or isothermal, etc.?

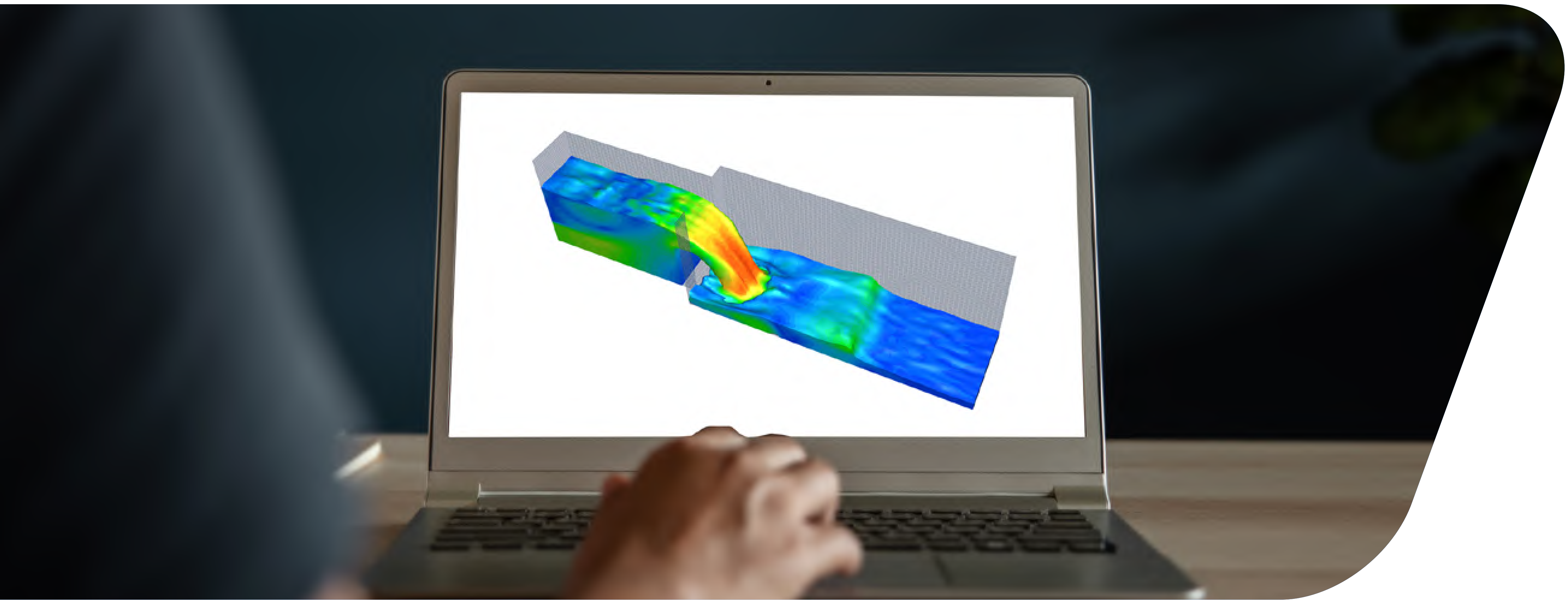
Specified boundary conditions and chosen appropriate modeling assumptions, the solver uses iterative techniques to improve the solution until “convergence” is successively attained.

The CFD program then iteratively solves the equations to convergence

Mathematically, convergence describes the limiting behavior of a series toward its limit. In CFD, the series is the flow field (values for velocity, pressures, etc.). The flow field reaches its limit when the velocity, pressure, etc., values stop changing from iteration to iteration.

In the separator world, multiphase modeling is key. There are three main multiphase models used in CFD: free-surface or volume of fluid (VOF); Eulerian-Lagrangian multiphase, often shortened to Lagrangian multiphase (LMP); and Eulerian multiphase (EMP).

In the analysis of separators, EMP can be used to model the complete vessel but is particularly effective in mixing regions where volume fractions exceed the limitations of LMP and tracking small droplets or bubbles with VOF is computationally expensive.



Case Study:
Three Phase (Multi-Phase)/ Transient CFD Analysis Flow inside the Separator Pressure Vessels

Three Phase (Multi-Phase)/Transient CFD Analysis Flow inside the Separator Pressure Vessels

Objective

To simulate transient three-phase flow inside the three-phase separator to observe phase separation inside the separator.

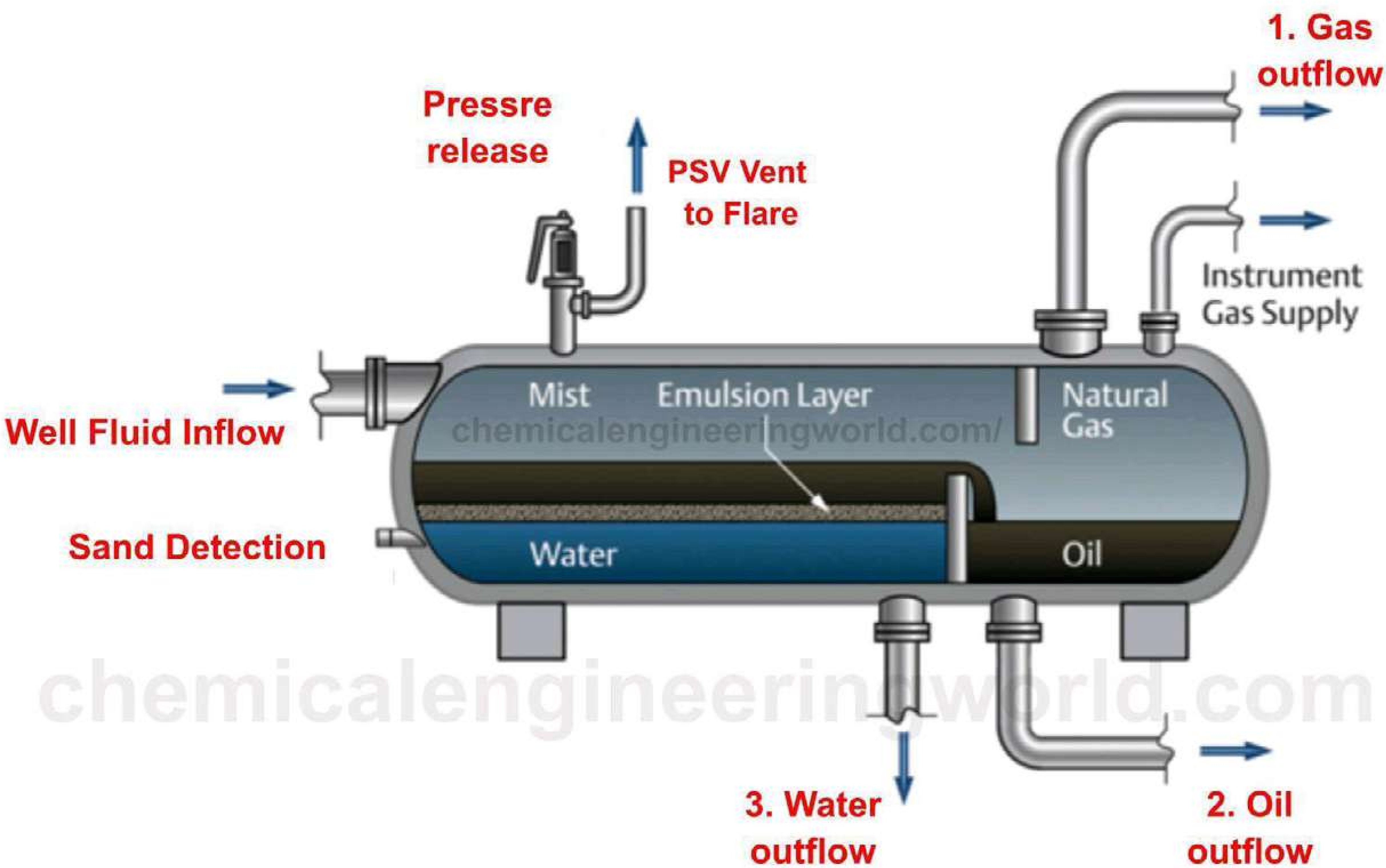
To check whether there is any mal distribution across the proposed inlet distributor design and also to compute the pressure acting on the inlet distributor walls and also on its supports.

To confirm that the outlet oil collector doesn't create upstream mal distribution through the electrostatic grids and also to compute the pressure acting on the oil outlet collector duct walls and on its supports.

Design Approach

The Population Balance Model (PBM) simulations have been initiated using a developed flow resulting from simulations with the Eulerian-Eulerian model for more rapid convergence of the solution. The inlet has been applied with fluid flow rate and fluid properties of phases. It is required to check the possibility of phase inversion at the inlet; therefore, a pipe length of 6 m upstream has been modelled. The inlet of the cyclonic device is modelled as per actual.

The porous media formulation has been used to model the cyclonic outlet device based on flow rate. Pressure drops across perforated baffle plates, and foam breakers are modelled using porous formulation.



Conclusion

From the CFD analysis, it was observed that the maldistribution across the inlet distributor opening is very small and almost equal flowrates are observed from each opening slot. However, some laterals with longer length have higher flowrates owing to increased number of slots across the lateral arm.

It can be concluded from the CFD analysis:

- A clear separation of liquid and gaseous phases is observed throughout the inlet pipe.
- Maximum Water in Oil (WIO) is calculated at 2.3% v/v,
- Most water droplets in Liquid HC are greater than 100 microns, and maximum Oil in Water (OIW) is calculated 243 ppmw.
- Time-averaged liquid carryover at the gas outlet is calculated as less than the 0.132 USG/ MMSCF.

Based on the obtained CFD results, we Mechartes also observed from the flow path developed inside the desalter that there are no large-scale recirculation or maldistribution inside the vessel.

With Mechartés expertise in the complex CFD and FEA domain in the Oil & Gas and Power sector, and the results/recommendations mentioned in the report the client was able to validate the design of its 3-phase separator.

We at Mechartés are focused on providing accurate simulation results with a professional engineering approach. Our biggest strength is our clients satisfaction with our work, our domain expertise, and team of young, passionate engineers who work as a close-knit unit.

Feel free to share your thoughts and observations at pcm@mechartes.com.

About Mechartes Services in Oil & Gas Sector

About Mechartes Services in Oil & Gas Sector

We at Mechatres are focussed on providing accurate simulation results with a professional and engineering approach. Our simulations represent the system closely and physics correctly at each parameter and step. With expertise in advanced numerical tools like Computational Fluid Dynamics (CFD) and Finite Element Method (FEM), our Team members have experience of more than 12+ years and have been successfully delivering projects in varied sectors:

- Single & Multiphase Separator CFD flow and FEA Analysis
- API 650 and API 620 Storage Tank Mechanical and Structural Design
- API 674 and API 618 based Pulsation analysis and Mechanical Vibration Study.
- Root Cause Analysis
- FIV, FIT, AND FIAV STUDY
- Piping Stress & Support design
- Surge and Slug Analysis
- Mixer and Agitator Design Analysis
- Maldistribution Study
- FEA for Pressure Vessel and Internals

You can find more case studies and other resources on our website.

Contact Us

Global Sales Office

Mechartes International FZC
PO Box 125396, Dubai, UAE.

+971 4 381 2060

+971 56 1156701 +971 56 1156702

srihari@mechartes.com sales@mechartes.com

Global Engineering Office

Mechartes Researchers Pvt. Ltd.
D-57, Sector 6, NOIDA, Uttar Pradesh, India - 201301

+91 880 017 7554

+91 120 454 0208 +91 120 413 1889

contact@mechartes.com fahad@mechartes.com

That was...

Multiphase CFD Simulation for Separators Pressure Vessel

Mechartés

SIMULATION EXPERTS