TENSYS Engineering • Analysis

CFD

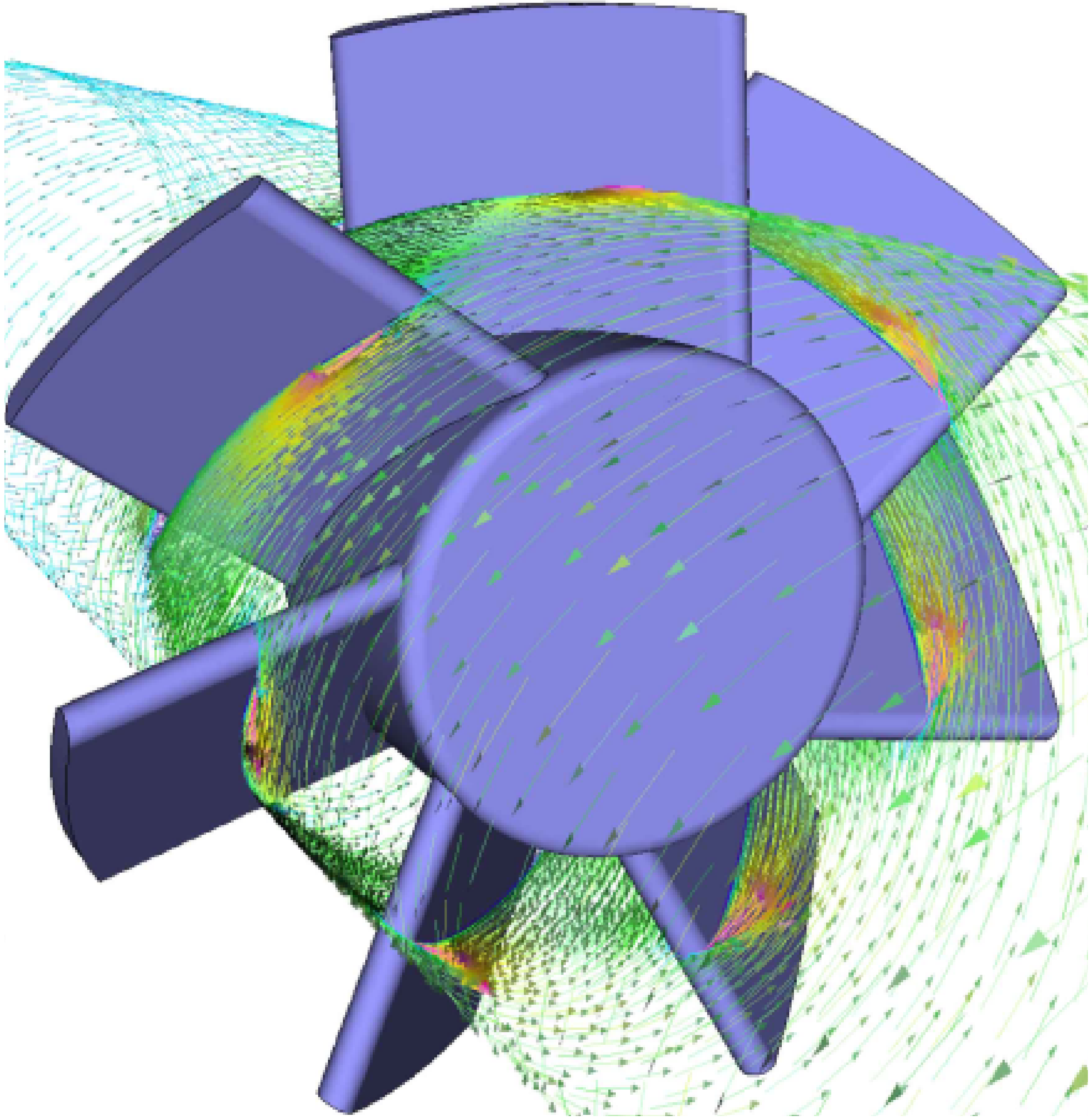
Tensys Dynamics provide consultancy services in the area of Fluid Mechanics and Computational Fluid Dynamics (CFD). We have many years experience of solving and delivering fluid flow based solutions across different industries saving our clients money and time. By leveraging our understanding and experience of fluid flow behaviour and the use of state-of-the-art simulation software, we create value for our clients far and above that which could be achieved using physical testing alone. See the links above for more on what can be achieved using CFD, and then contact us for an informal review of your problem. We work in the following areas:

Building Aerodynamics: Pedestrian Wind Environment Assessments, Passive Ventilation Intake/Outlet Placement, Wind Turbine Siting, External Pollutant Dispersal.

Ventilation: HVAC performance, Air Quality and Thermal Comfort Assessments, Contaminant Migration, Condensation Formation.

Industrial Design: Energy Conversion, Heat Transfer and Cooling, Separation Processes, Minimisation of Pressure Losses in Ducts.

Vehicle Aerodynamics: Calculation of Aerodynamic Loads and Deflection of Airship Envelopes, Sails and other Tensile Fabrics, Lift and Drag optimisation of Vehicle Bodies.



TENSYS Engineering • Analysis

Building Aerodynamics

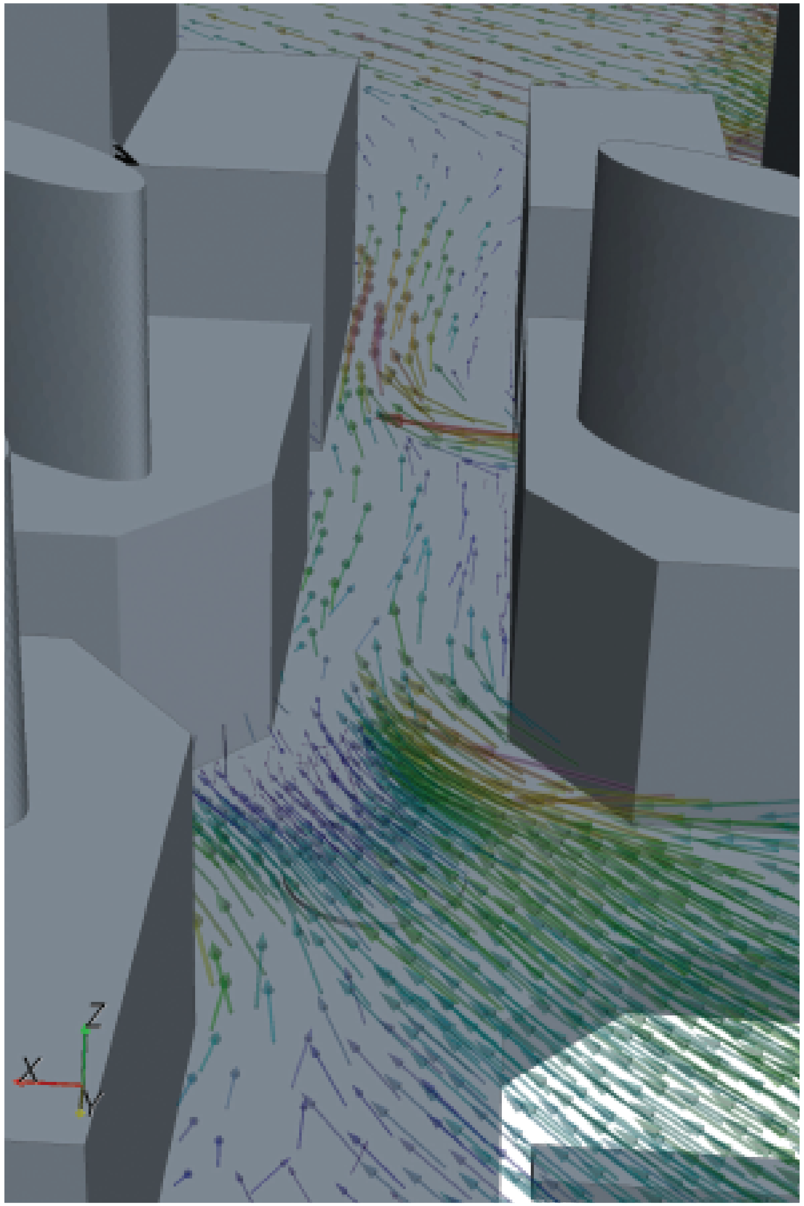
Evaluating buildings' aerodynamic properties is becoming an increasingly popular activity due to the large cost of these structures and the relative ease with which simulations can now be carried out. Performing a CFD analysis in the early stages of a project can reduce overall costs, cut the critical path and eliminate expensive mistakes downstream. Calculations are performed at full scale hence removing any uncertainty due to scaling effects.

Atmospheric flows around buildings for pedestrian wind environment assessments
A simulation of the air flow between a potential configurations of buildings can give an early warning of inadvertently increasing wind speed and turbulence at pedestrian level. Where a problem already exists, we can perform retrospective analysis and design comfort enhancing solutions, e.g. fabric barriers.

Placement of ventilation system intakes and outlets
Passive and semi passive ventilation systems require accurate placement of intakes and outlets in order to maximise the pressure differential driving force hence reducing energy usage, cost and Carbon emissions. This type of problem is well suited to CFD analysis due to the detailed resolution of the surface pressure distribution.

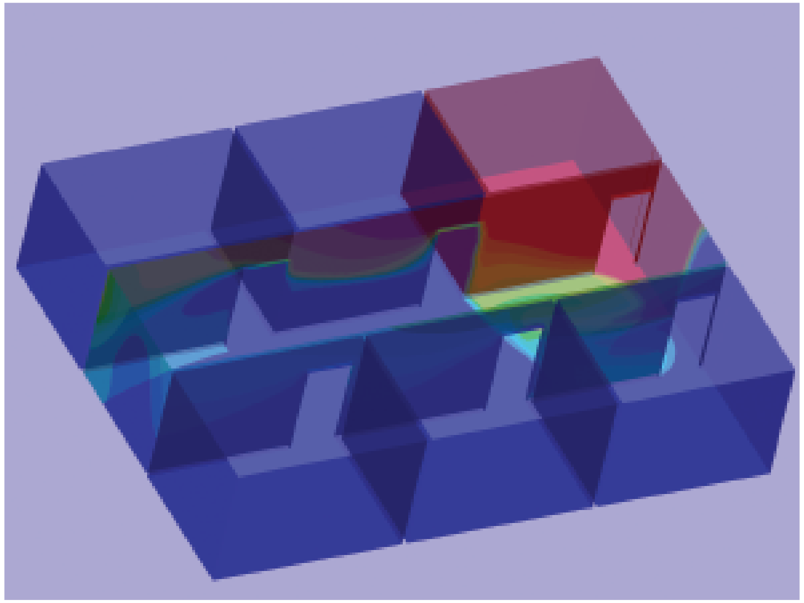
Wind turbine siting
The exact placement of building mounted or building integrated wind turbines can be critical to the success of a project since the wind speed varies considerably as it passes over a building. A CFD simulation for the predominant wind directions will be able to locate the best position for the turbines and maximise energy yield.

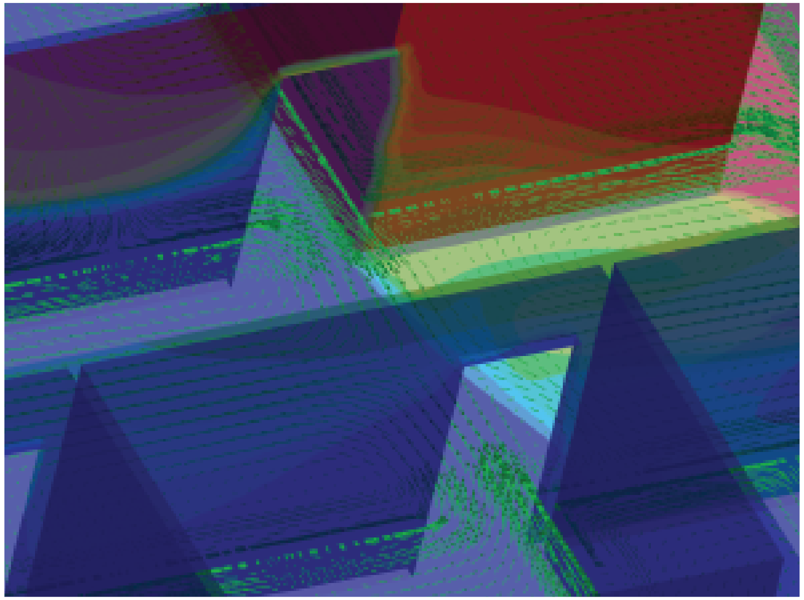
External pollutant dispersal
Environmental risk assessments due to accidental release of pollutants from buildings are difficult to recreate due to scaling problems and the high cost of performing the experiments. An alternative is to simulate the release of a pollutant at full scale using CFD. This is significantly cheaper and provides more detailed results.



TENSYS
Engineering • Analysis

Ventilation





Building services are estimated to account for 40-50 % of primary energy consumption in industrialised countries. With increasing environmental awareness and economic pressures on energy consumption there is both good reason and plenty of scope for improvement. Air flow, heat transfer, occupant comfort and contaminant migration can be calculated accurately using a full CFD simulation which takes into account the major geometric entities, inlet, outlet positions and heat sources.

HVAC performance, air quality and thermal comfort studies

In order to evaluate the internal environmental quality of a building, air velocities, temperatures, turbulence intensity and air quality must all be taken into account. In terms of occupant comfort, low air speed and a narrow range of temperature variation are required along with the opposing constraint of sufficient ventilation, which demands higher air speeds. CFD analysis is an effective method to assess the various design solutions and in most circumstances is the only tool which can practically achieve this. Draughts, stagnant regions, comfort performance data can be easily extracted from the results.

Contaminant migration in enclosed spaces

In laboratories, clean rooms, hospitals, etc. the migration of particulates and/or gaseous containments can be traced and prevention measures virtually introduced and tested. Concentrations and migration times fall naturally out of the analysis.

Condensation formation and moisture movement

By modeling the concentration of water vapour and the air temperature and hence calculating the relative humidity of the air at any location. The risk of condensation forming can be calculated from the dew point and the building's surface temperatures.

TENSYS
Engineering • Analysis

Industrial Design

The design of commercial products and industrial machines regularly involve optimising a fluid mechanical effect of some kind. Broadly these can be divided according to their function, ie. energy conversion, heat transfer and cooling, separation processes and minimising pressure losses. The objective is to satisfy the design constraints of, for example, efficiency, cost, space and weight.

Energy Conversion

There is an increasing interest in the development of alternative low carbon emitting, renewable energy sources. Wave energy converters, marine current turbines, wind turbines, solar collectors, low grade heat conversion and biomass burners all utilize fluid mechanical effects, which can be modelled and improved using CFD analysis. Tensys Dynamics is interested in cooperating positively with companies involved in the development of these technologies.

Heat transfer and cooling

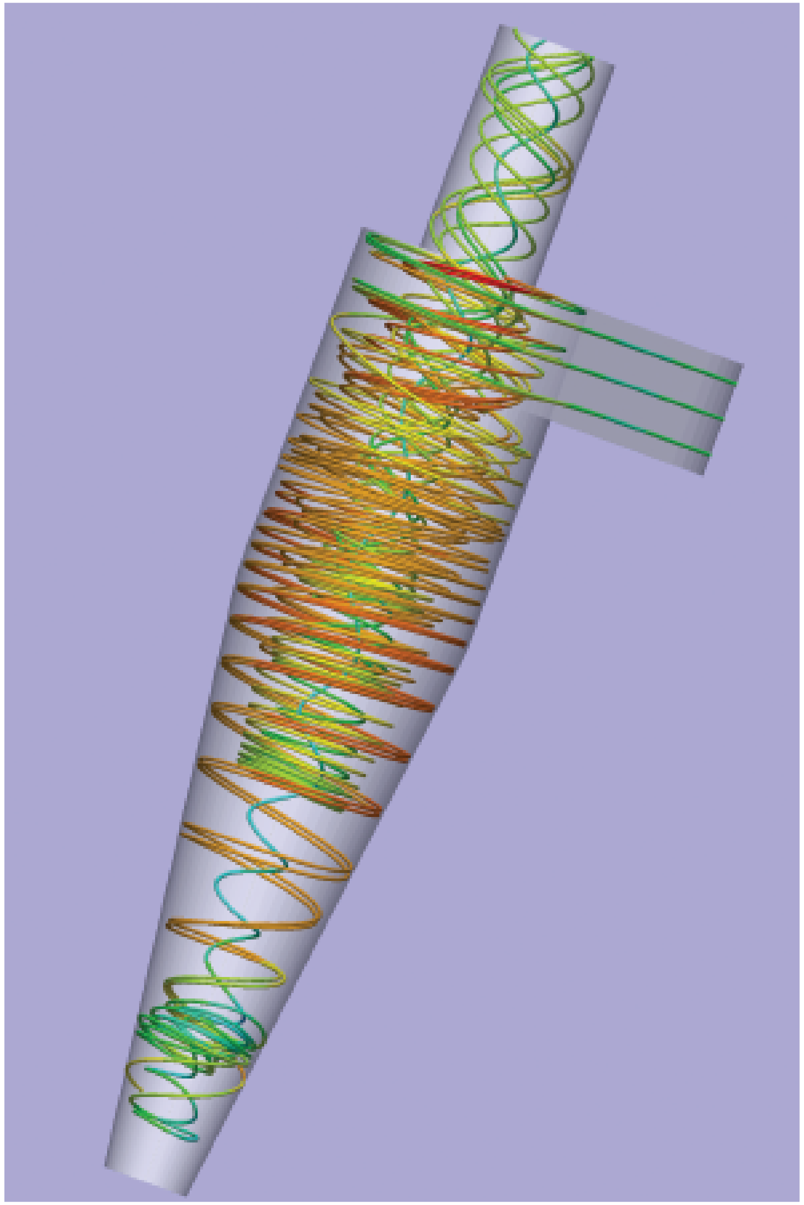
Computational Fluid Dynamics analysis is now a well-established method in the design of heat transfer and cooling applications. Typically the fluid flow is confined by walls and is consequently more stable and predictable than unconfined flows. Application areas include heat exchangers, HVAC and condenser optimisation, heat pumps and the cooling of electronic components.

Cyclone separators

The separation of different density phases from each other in a fluid stream can be achieved using devices which generate large centrifugal forces, for example cyclone separators. Tensys Dynamics have significant expertise in this area and can advise on all aspect of separation system development. Applications include domestic appliances, hazardous emissions control and off shore processing.

Minimising Pressure losses

Reducing the difference between entry and exit pressures through pipes, ducts, orifices, manifolds and plenums requires pin pointing the offending pressure loss and managing the fluid flow through its path such that the entire loss is minimised. CFD is an ideal tool for this purpose, since it can accurately locate the source of the losses and allows us to virtually prototype design solutions quickly and reliably.



TENSYS
Engineering • Analysis

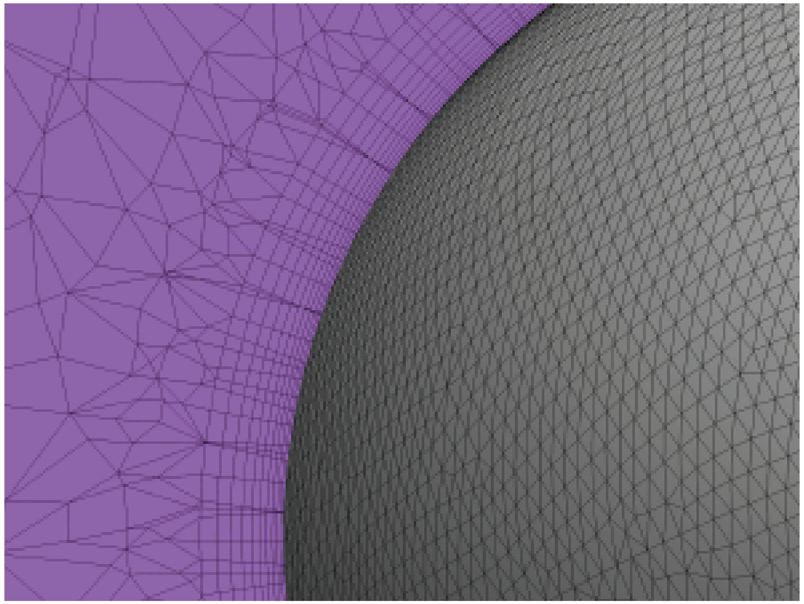
Vehicle Aerodynamics

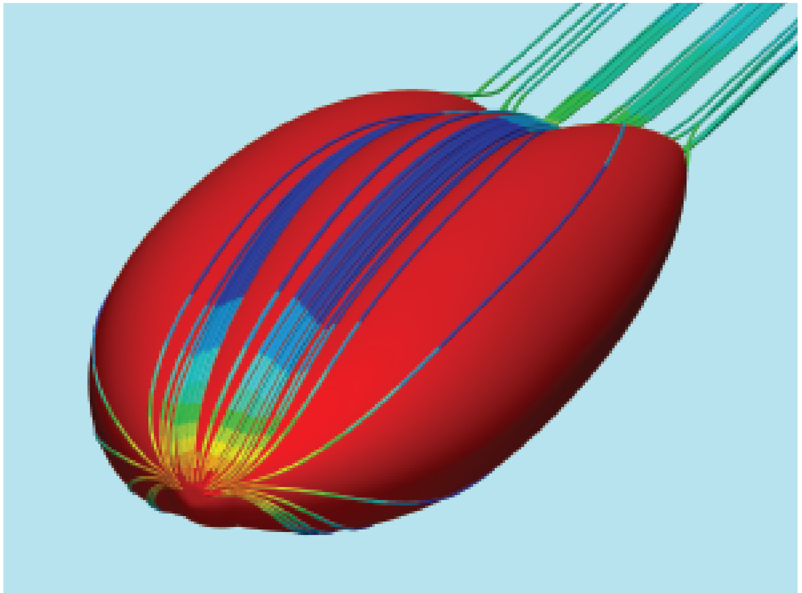
Computational Fluid Dynamics was developed in the 1960's in the aerospace industry primarily to predict the aerodynamic forces on aircraft, consequently this area of modeling is well understood and numerous validation exercises have been carried out. Minimising the drag force and increasing stability are common objectives. Applications are found in the aerospace, automotive, and marine industries and increasingly in sports science.

Airship Design
Whilst the fluid flow structure on a conventional airship envelope is relatively streamlined at zero angle of attack, once real flight values are attained, the flow becomes highly complex with strong pressure gradients and cross flow separation. The lift, drag and pitching moments then become more sensitive to the geometric details of the supporting structure and location of propulsion units, effecting operating efficiency and manoeuvrability. Tensys have an established expertise in the design of airships and their envelopes for conventional and hybrid forms, we can now offer complementary services in the calculation of lift and drag forces and additional loads on the envelope. This is possible by linking the CFD calculation with our inTENS finite element software.

Aerodynamic Loads on Tensile Fabrics
Coupling our inTENS finite element software to the CFD calculation allows us to calculate the fluid structure interaction (FSI) and hence the resulting deflection due to aerodynamic forces on sails etc. From the equilibrium form the resultant propulsive forces can then be extracted for different initial configurations. Conversely the flight and braking characteristics of para gliders , parachutes and hang gliders can be studied. Tensys have additional expertise in this area through their extensive experience with the analysis of fabric stresses and deflection under aerodynamic load.

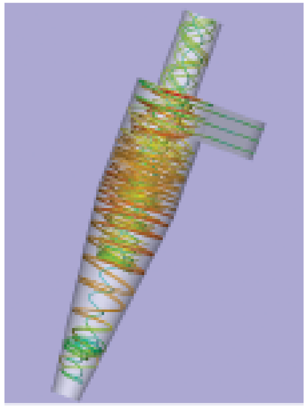
Lift and Drag optimisation
The accurate calculation of both lift and drag using CFD requires considerable expertise. As does locating the causes of excessive drag and designing a solution. Applications are numerous and include automotives, fairings, motorbikes, racing cycles, boat hulls, aerofoil and hydrofoil.

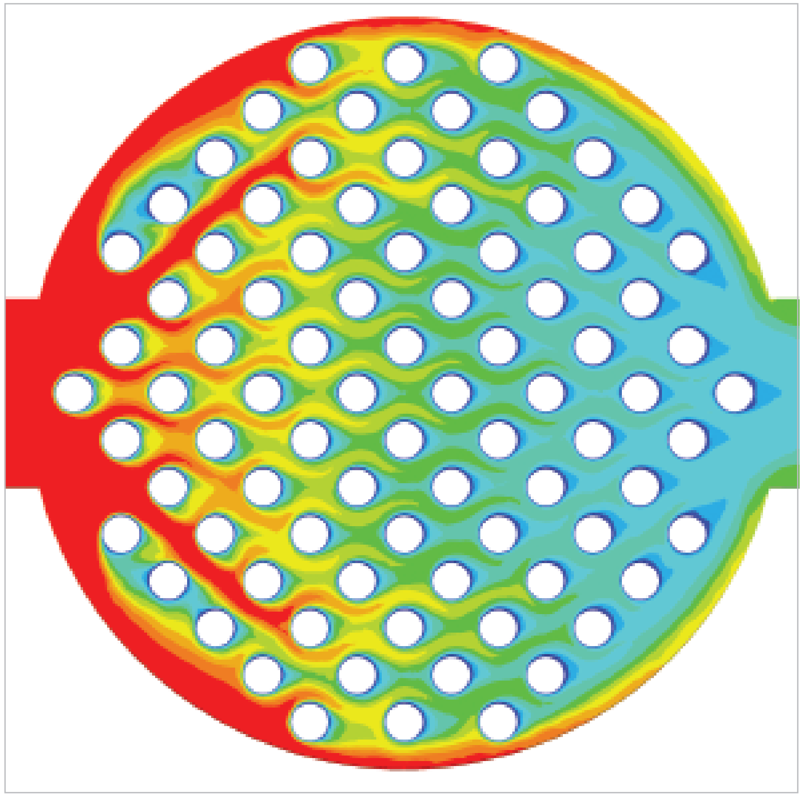




TENSYS Engineering • Analysis

How We Work





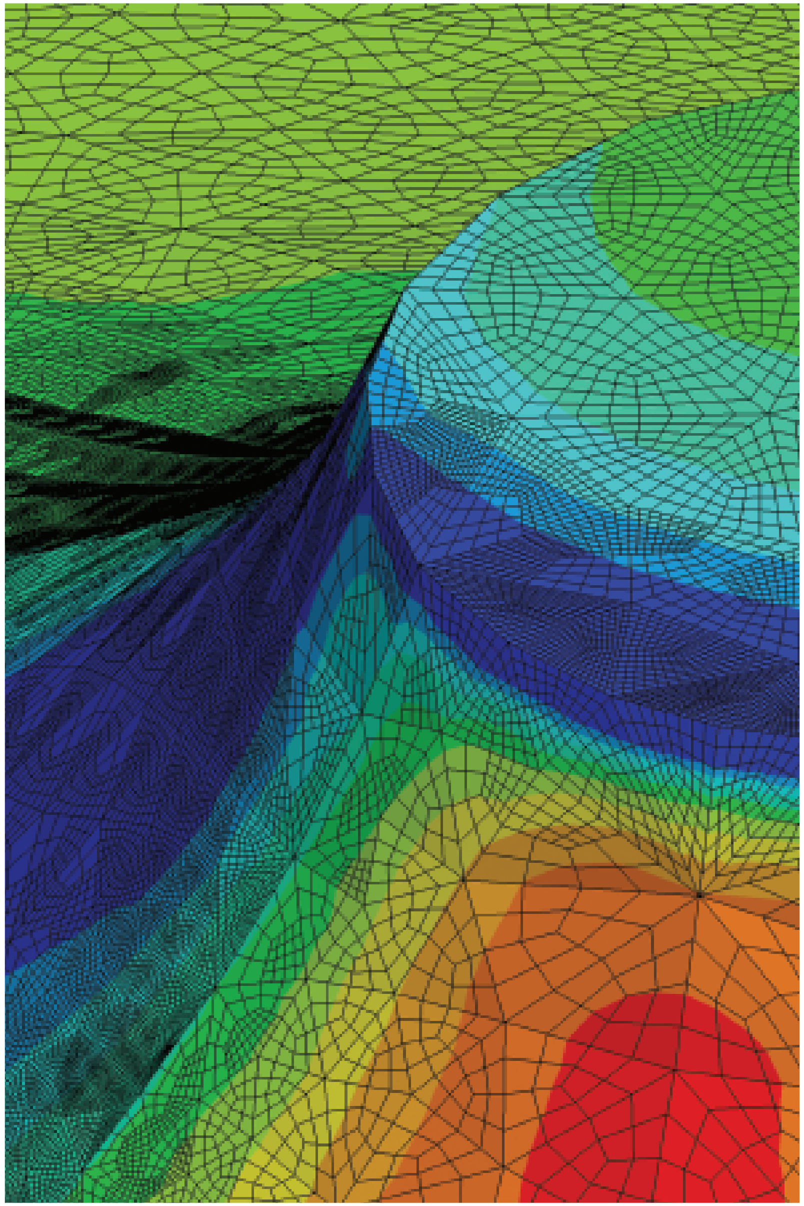
Our fluid flow consulting services are targeted to serve engineers working in SME's in industries that require analysis and optimisation of fluid mechanical effects. We use CD-Adapco's suite of Computational Fluid Dynamics (CFD) simulation software including STAR-CD and the state-of-the-art computational continuum mechanics code STAR-CCM+. We are currently working towards the incorporation of our CFD activities into the Tensys quality management system which is accredited to BS EN ISO 9001 and can work either on a fixed price or a time spent basis and to agreed timescales.

Our aim is to integrate into your engineering design process by providing an expert resource feeding back results in a timely manner. In some cases only a quick analysis employing simplifying assumptions might be appropriate, perhaps using only hand calculations, whilst in others a very realistic analysis may be required where for example the results are compared with experiments. Our responsibilities are defined by negotiation with the client at the start of the project and range from merely providing results for integrated quantities of engineering interest to taking on the fluid mechanical design from start to finish. Contact us for an informal review of your problem.

Tensys recognises the importance of validating the CFD method for different application areas. As part of our ongoing validation programme we seek out experimental measurements for cases similar to our clients and perform our own analysis in order to assess the accuracy of the CFD modeling approach for this situation. In addition we can refer to validation studies available in the open literature. This we believe is fundamental to giving our clients confidence in us and in CFD as a technique.

TENSYS
Engineering · Analysis

CFD Overview



Backgound

Computational Fluid Dynamics (CFD) is a method of calculating the velocity, pressure and temperature fields in a region of space occupied by a fluid. Essentially the space is divided up into a large number of small volumes (cells) defined by a 3 dimensional grid. The non linear partial differential equations governing the conservation of mass, momentum and energy in the fluid are then solved by a numerical method on a computer. Because the region of space is being artificially separated from its surroundings it is necessary to define the conditions the fluid experiences on the boundaries of the region (the boundary conditions) since this will influence the flow of fluid inside. The more accurately the boundary conditions and surrounding geometry can be specified the more accurate will be the calculation inside the region.

The process

In practical terms the process usually starts with a CAD drawing of the component, building etc. The drawing is usually supplied by the client is a generic CAD format such as iges, parasolid, iges, step, stl, dxf, etc. A computational grid is then generated inside the region with the highest concentration of cells being placed close to solid boundaries and where the gradient of key variables are likely to be large. Boundary conditions are then defined. These could be as simple as an uniform inlet velocity or pressure or more complicated conditions such as an atmospheric boundary layer over rough terrain. The calculation is further defined by any particular models required, eg any porous media characteristics, transient analysis or particular turbulence models. The properties of the fluid itself must also be defined ie. density, viscosity and specific heat capacity. Finally the parameters relating to the numerical method are selected and the calculation is started. Typically the solution will converge in a few hours, after which the quantities of interest can be extracted from the raw results using sophisticated post processing software. Further sensitivity checks are then carried out, by repeating the calculation with different parameters to ensure that for example, the grid is sufficiently refined to capture the gradients in the flow field.

continued

CFD Overview (continued)

Turbulence

In all but the simplest of cases the fluid flow will be turbulent, this means that the fluid is changing direction and pressure at each point in the region rapidly in time. In theory it is possible to calculate these rapid fluctuations but in practice the computing power required to do so is not likely to be available for at least 30 years. A practical solution to this dilemma is to attempt to calculate the average velocity, pressure and temperature, at each point by making assumptions about how the fluid is likely to fluctuate. This introduces assumptions, which are correct to different extents depending on the particular regions geometry, boundary conditions and the fluid itself. These assumptions effectively define what is known as the turbulence model and contribute to any mismatch between the CFD calculation and experimental measurements made on an analogous system. Consequently it is important for us to have an estimate of the CFD model's accuracy (validation) for each application area. Usually this is known already or else achieved by us simply examining previous studies carried out in the open literature, or by carrying out our own validation study.

Why use CFD

CFD is one of several CAE tools in common use today throughout industry and academia, Finite Element Analysis (FEA) uses very similar techniques for the analysis of stress and vibration in solid materials. Both methodologies give the engineer the opportunity to rapidly assess different design scenarios without having to resort to prototype manufacture and testing. In addition there are circumstances where it may be impractical or even impossible to create a suitable experiment, for example where scaling arguments cannot be applied, or intrusive measurements are not possible. Advanced post processing tools enable the results from a CFD calculation to provide detailed insight into how the fluid is behaving, for example streamlines, particle tracks, velocity vectors and pressure contour plots. In addition integrated quantities such as normal and shear forces or surface averaged values are naturally extracted from the results providing the engineer with information to measure performance and inform how design changes might be made. As computing power has increased and engineers have gained confidence in CFD its use has become more widespread and more integrated into the design cycle. This trend is set to continue.