PHOENICS Newsletter

CHAM

Dear Reader

It is that time of year when trees turn green, flowers bloom, the sun shines and the mind turns to new activities and possibilities.

At CHAM it is the time when minds turn to working on, and completing, the new, annual, version of PHOENICS for distribution to all maintained users and for provision to all new clients.

Work on PHOENICS-2022 is, thus, underway. It will contain Items including an extension to VOF; an extended range of Non-Newtonian fluids; Sutherland's Law of thermal conductivity; a revision of Unstructured PHOENICS, and other items. A more detailed account appears on page 2 of this Newsletter. If you would like further information regarding content, or availability, please contact sales@cham.co.uk.

PHOENICS-On-The-Cloud is increasing in popularity as a cost-effective way to run large cases on the powerful multi-core systems offered by the Microsoft Azure marketplace. You can explore this option via: http://www.cham.co.uk/_docs/pdfs/phoenics_docs/PHOENICS-OTC2021.pdf.

Our website Testimonials section is expanding. If you have worked with our User Support, Consultancy, or Software Teams we would like to hear your experiences. Please email website@cham.co.uk.

We are looking for a new member of staff to join our Wimbledon-based team. See: https://www.cham.co.uk/careers_description_Consultancy_Engineer.php. If you would like to consider this opportunity, or know of anyone who may be interested (and who is legally entitled to work in the UK), please send CV, and cover letter, to hr@cham.co.uk. Thank you.

Kind Regards and, slightly prematurely, Happy Easter.

Colleen Spalding
Managing Director

TABLE OF CONTENTS

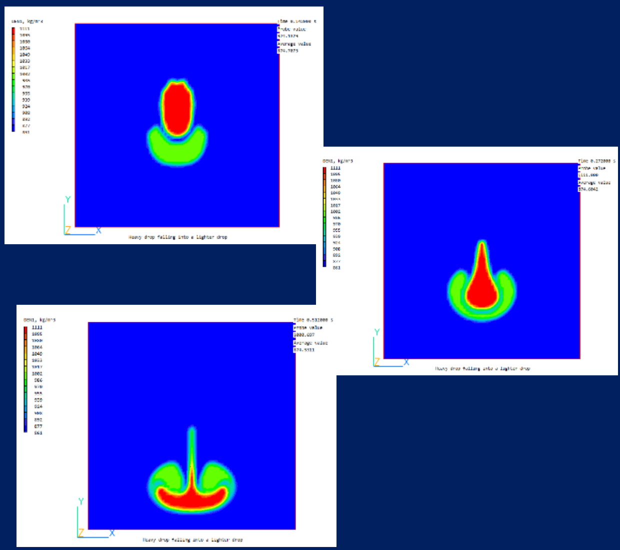
Concentration, Heat and Momentum Limited (CHAM)
Bakery House, 40 High Street, Wimbledon Village, London, SW19 5AU, England
Tel: +44 (0)20 8947 7651 Email: phoenics@cham.co.uk Web: www.cham.co.uk

Spring 2022

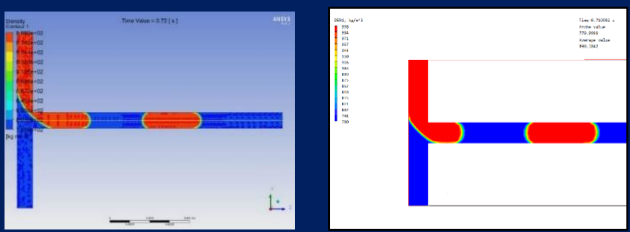
Some Features of PHOENICS-2022

3-PHASE VOF

VOF now allows for 3 distinct phases, gas/liquid/gas, liquid/gas/liquid, liquid/liquid/liquid or gas/gas/gas. A heavy drop falling into a lighter one:



The contact angle calculation has been improved. Comparison with Ansys for water / kerosene slug flow:

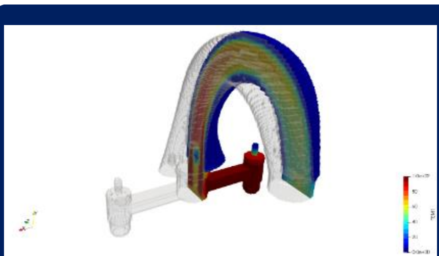


Extended range of Non-Newtonian fluids

Seven additional non-Newtonian models have been coded into PHOENICS allowing for simulation of an extensive range of fluids, including blood, clay, foods, greases, mud, polymers, sewage sludge, and slurries.

Unstructured PHOENICS (USP)

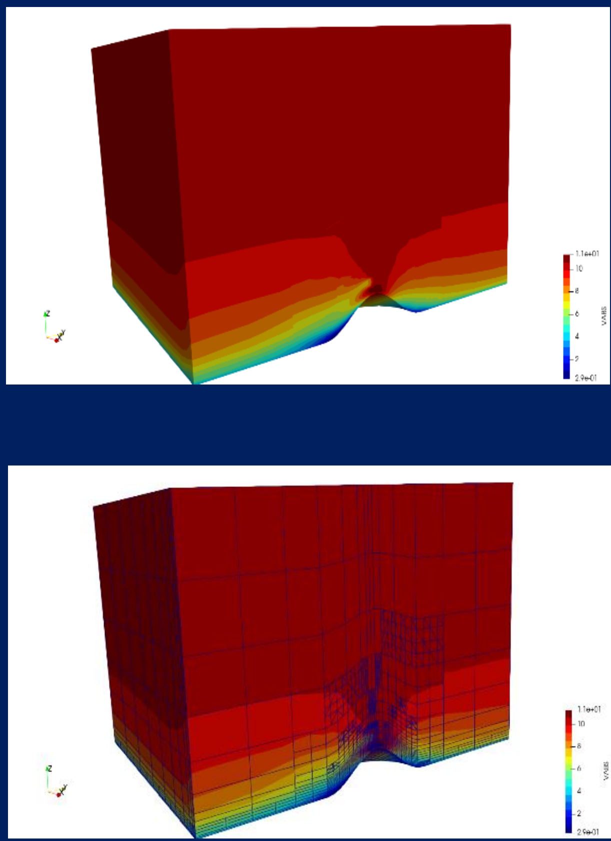
USP will include (a) display of USP mesh in VRE & USP solutions in VRV, (b) allowance for parallel operation, and (c) additional models and comfort indices.



Flow Through a Tap

USP-BFC for unstructured terrain model

A USP version was created for WindSim using a terrain-following BFC grid as the unstructured mesh starting point. The resulting grid can incorporate local refinements near ground plane and around wind turbines (represented as actuator disks using InForm). The code runs in parallel.



Flow over a Cosine Hill with Turbine and local refinement

IPSA Extension to include Kinetic Theory of Granular Flow (KTGF)

IPSA is extended to allow particle-particle interactions by implementing several models for the solids stress tensor that are all based on the kinetic theory of granular flow. The kinetic theory of granular flow involves solving an equation for the granular temperature, which represents the kinetic energy of the fluctuating particle velocity field. The IPSA-KTGF model extends PHOENICS-2022 to handle more realistically solid-fluid applications like dense-regime pneumatic lines, riser reactors, and fluidised-bed reactors.

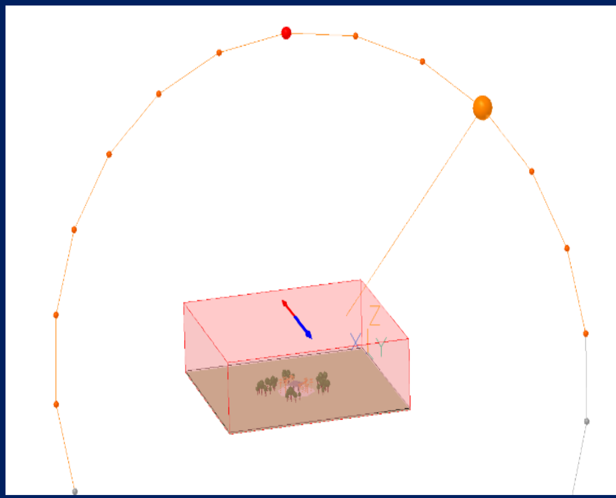
Pedestrian comfort beneath metal canopy with foliage surround, by Peter Spalding and Andrew Carmichael, CHAM, London, October 2021

CHAM was asked by Architectural Studio designer, Elias Sanchez to simulate the effect of solar shading upon pedestrians occupying an area beneath a canopy with an intricate metal roof design situated within a park in Culiacán, Sinaloa, Mexico, where summer temperatures often exceed 35°C with high levels of relative humidity. The client's interest focused upon the solar incidence and shadow behaviour based upon meteorological data. The requirement was to determine thermal comfort experienced by pedestrians passing underneath, or seated around, an open plaza consisting of a partially-open metal roof canopy built upon a stone base and grass surround.

The purpose was to ascertain the occupants' overall comfort, taking into consideration the prevailing environmental conditions - i.e. wind, temperature and relative humidity - whilst including the added effect of shading (and potential cooling) from surrounding trees and novel artificial star-shaped lamp posts. The design was supplied in CAD form as a 3D model in 3DS format and imported directly into PHOENICS/FLAIR.

CAD model displayed in PHOENICS-VR with shading activated

The case made use of several features inbuilt in FLAIR to simplify, greatly, the process of problem definition. The SUN object determines appropriate solar load and angles and calculates areas of shading that can then be used, in conjunction with, IMMERSOL, to determine radiative heat transfer from all surfaces. The WIND object controls the setup of the atmospheric boundary layer, temperature and humidity levels. The trees are modelled using a FOLIAGE object, which implements a drag model to these areas, applies correct shading and a source of "coolth" as, naturally, the trees will extract some heat from the surrounding air.



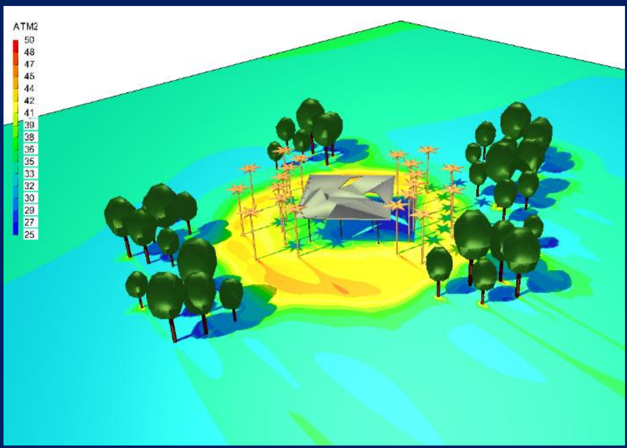
Sun & wind strength and direction added via the Sun & Wind objects

The capability of PHOENICS/FLAIR to interface with third party software products (such as EnergyPlus™) to extract meteorological data for a specific location, was not used on this occasion; such data was added manually via the GUI.

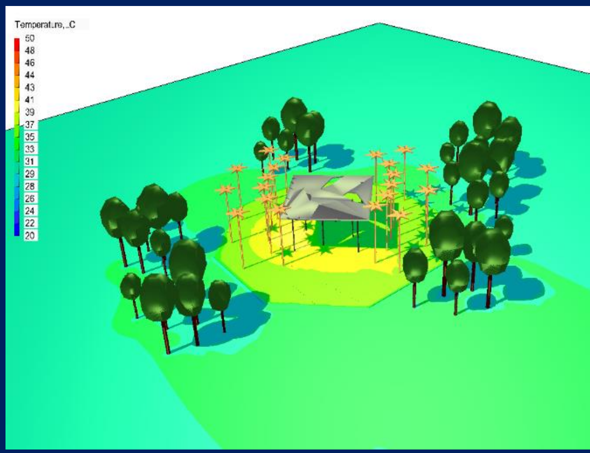
All the variables come together to answer the client's original question: How effective is their roof design at providing a comfortable environment for people in the park? This can be measured directly using one of FLAIR's available comfort indices, in this case the Apparent Temperature, which takes into account local air temperature and velocity, radiant temperature from the sun plus the metal roof, the amount of radiation absorbed by a person depending on clothing and humidity level to yield the "Feels like" temperature so often quoted in weather reports.

Solar radiation

In order to make a true comparison of the results of the simulation with measured data, it is usual to run the CFD model transiently to capture heating and cooling processes over time. For the purpose of canopy design however, this case can be run as a steady-state simulation to represent the meteorological effects taken as a snapshot in time, employing a 'solar absorption' factor to ascertain a reasonable trend.



Apparent temperature [CHAM Oct 2021 revision] @ 1.8m (with shading)



Ground temperature (with shading)

PHOENICS/FLAIR can now be accessed 'on the cloud' via the Microsoft Azure Marketplace, allowing clients the full capability of urban CFD modelling using high-performance processing power and without the need for either software download or licence purchase. See: http://www.cham.co.uk/_docs/pdfs/phoenics_docs/PHOENICS-OTC2021.pdf

Flow through a Jet Pump

By Timothy Brauner, Andrew Carmichael, and Michael Malin,
CHAM London, February 2022

Introduction

Jet pumps are used in many areas of engineering. In various different configurations they can be used to transport gases, liquids or solids suspended in a fluid, like particles in air. The driving, or motive, fluid can be a gas or a liquid. (When a gas is used the device is generally referred to as an ejector rather than a jet pump.)

Jet pumps have no moving parts and consist simply of a nozzle, a suction/receiving pipe and mixing chamber/throat into which the product to be moved is drawn. They are powered by a supply of pressurized fluid, often from a conventional centrifugal pump. The motive fluid passes through a nozzle converting pressure energy into kinetic energy, creating a high-speed, low-pressure jet. This low pressure at the nozzle exit and in the throat induces flow of the product through a shroud or suction duct into the mixing chamber/throat. The streams combine within the mixing chamber and discharge together against the pressure head required to drive the pipeline.

Figure 1 shows a slice through the centreline of the geometry of a typical jet pump.

On the left of the figure the motive fluid's inlet is indicated by the red arrow.

Surrounding it is the suction/receiving pipe with its inflow indicated by the two blue arrows. Downstream of the nozzle exit the geometry contracts to form the mixing chamber/throat. Further downstream, motive fluid and product leave the jet pump through a diverging section/diffuser, symbolised by the purple arrow on the right of the figure.

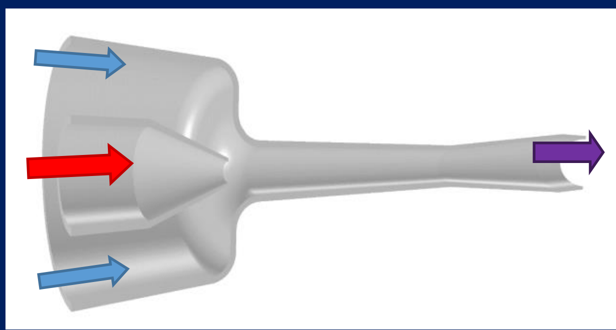


Figure 1 – Cross sectional view of an example jet pump geometry. Arrows indicate in- and outflows.

Some example applications of jet pumps are creating offshore trenches for pipes and cables, silt removal from settling ponds, enhancing flow rate of a smaller pump, transporting particles, spray painting or placing of other materials with air as the motive fluid.

Problem Specification

PHOENICS has been used to investigate characteristics of a jet pump for a set of prescribed boundary conditions; results were compared with experimental data reported in [1].

Primary geometrical features of interest in studies are the nozzle-to-throat area ratio, throat length and nozzle-to-throat distance. Flow characteristics of interest are upstream and downstream pressures, and the ratio of motive fluid and product fluid flow rates. In the setups considered, the motive and product fluids are both water. In the numerical setup the pressures are prescribed at the inlets of the motive and secondary fluids, with the respective inlet velocities deduced from the simulation, a pressure is also prescribed at the outlet. Simulations are run in two-dimensional, axisymmetric, cylindrical-polar coordinates using S-PARSOL to handle cut cells.

Results and Discussion

Following the setup of [1], three boundary conditions were simulated, see Table 1, and the results are summarised in Figure 2. The mass flow ratio, $M_r = \dot{m}_s/\dot{m}_p$, and pressure ratio, $P_r = (p_{dt} - p_{st})/(p_{pt} - p_{dt})$, are used to define the efficiency of the jet pump as $\eta = M_r P_r$. The subscripts denote r: ratio, p: primary (motive), s: secondary (entrained), d: downstream and t: total. In Figure 2, the green squares represent the experimental data of [1] and the black lines are trend lines fitted to this data. The blue diamonds represent the three boundary conditions simulated in PHOENICS, all of which match the experimental data well.

Case Number in [1]	P_{pt}	P_{st}	P_d
[#]	[Pa]	[Pa]	[Pa]
3	588173	83325	204890
7	333118	83325	130679
12	233867	119825	135859

Table 1 – Simulated boundary conditions. P_{pt}: total pressure at the inlet of the primary/motive fluid, P_{st}: total pressure of the secondary / entrained fluid, P_d: static pressure at the outlet.

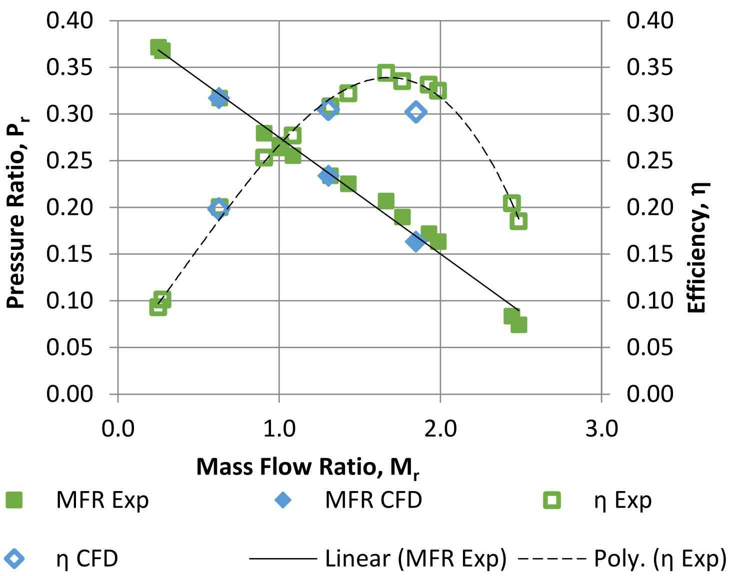


Figure 2 – Mass Flow Ratios (MFR) against Pressure Ratios and Efficiency. Experimental data represented by green squares with a linear best-fit line (-) for MRF and a third order polynomial best-fit dashed line (--) for Efficiency. PHOENICS CFD derived data is represented by blue diamonds.

Figures 3 and 4 show surface contours of velocity magnitude and total pressure for the 12th set of boundary conditions in [1], corresponding to the rightmost CFD data point in Figure 2.

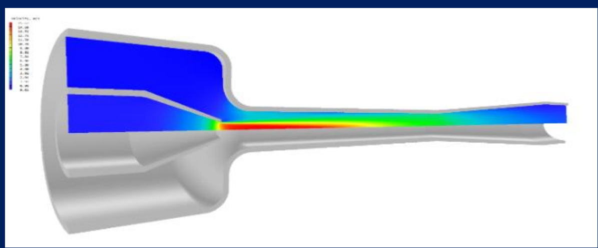


Figure 3 – Example surface contour of Velocity Magnitude

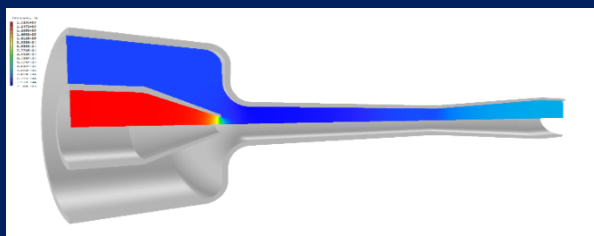


Figure 4 – Surface contour of Total Pressure relative to an ambient pressure of 101325 Pa.

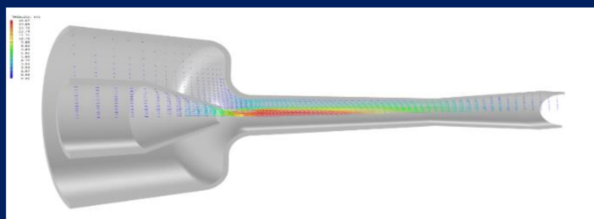


Figure 5 – Velocity vectors coloured by velocity magnitude.

Conclusion

The results produced by PHOENICS match the experimental data well, and one might now consider further simulations to investigate the sensitivity of the solutions to mesh or, similar to [1], the effect of using different turbulence closure models. The successful validation of PHOENICS for this case indicates that it can now be used to aid and optimize the design of liquid jet pumps, determining the effect of various housing shapes on overall flow, as well as the effect of different feeder flow rates with a view to determining the overall mass flow rate of the pump. Advanced calculations that account for multiple phases, for example if the motive flow was air and the fluid to be moved was water, could also be performed. PHOENICS can now be accessed on the cloud (PHOENICS-OTC) in the Azure Marketplace, providing clients with the full capabilities of PHOENICS for CFD modelling, coupled with high performance computational power without the need to purchase a full software licence.

References

[1] Aldas, Kemal & Yapıcı, R. (2014). Investigation of Effects of Scale and Surface Roughness on Efficiency of Water Jet Pumps Using CFD. Engineering Applications of Computational Fluid Mechanics. 8. 14-25. 10.1080/19942060.2014.11015494.

Acknowledgements

Professor Nikos Markatos and his student Georgios Gkoulionis of the National Technical University of Athens, Greece for supplying their original PHOENICS case files, which were then refined and extended to carry out the simulations reported in this article. Readers may be interested to know that during the early days of CHAM, Nikos worked for many years with Professor Brian Spalding pioneering the development and use of CFD technology across a very wide range of engineering and environmental applications.

Vehicle Aerodynamics: Flow Around the Ahmed Body

By Shakil Ahmed and Timothy Brauner, CHAM London, January 2022

Introduction

We at CHAM have prepared a benchmark case study for vehicular aerodynamics using an Ahmed Body to help our clients who use PHOENICS to get a running start when building CFD cases.

The Ahmed Body is a simplified vehicle design that allows for easy standardization of tests results between CFD solvers and experimental results. In this case study we computed the drag coefficient and analysed the flow behaviour in the areas surrounding the Ahmed body and compare them with experimental results.

For this case study we used a moderate grid resolution to obtain a sufficiently accurate drag coefficient value for the Ahmed body.

CAD Setup

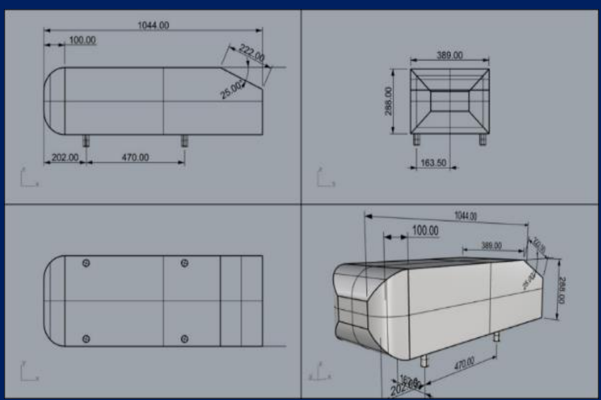


Figure 1 – size and dimension of the Ahmed Body in millimetre

Figure 1 shows the CAD model used for this study. You can find this CAD model available for download here.

The slant angle of the rear of the Ahmed body used for this specific case is 25 degrees.

Case Setup:

We used a domain size large enough to allow the flow to develop appropriately. In this tutorial we use approximately:

- 10 car lengths in the direction of the flow (X axis)
- 15 car widths in the direction normal to the flow (Y axis)
- 15 car heights in the vertical direction (Z axis)

This results in a domain that is:

- 10m in X
- 6m in Y
- 5m in Z

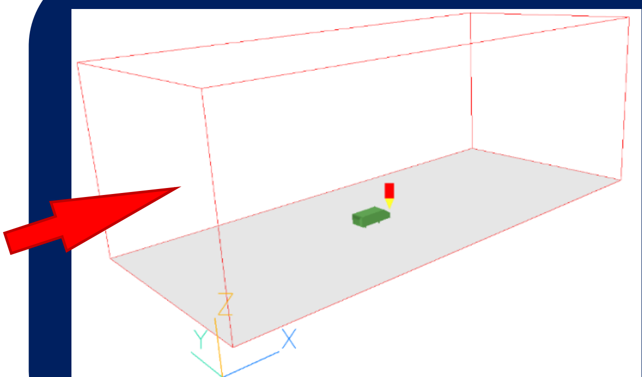


Figure 2 – overall domain bounding box

Boundary Conditions

An inlet velocity of 10 m/s is set for the X-min domain face. The turbulence intensity should be set to something low or off (User-set defaults) to mimic the effect of the vehicle driving into quiescent air.

An open boundary is set for the X-max domain face.

The ground plane also has a velocity of 10m/s to simulate the car moving relative to the ground.

This produces a Reynolds number of approximately 675,000, based on the inlet velocity, length of the Ahmed Body and properties of standard atmospheric air.

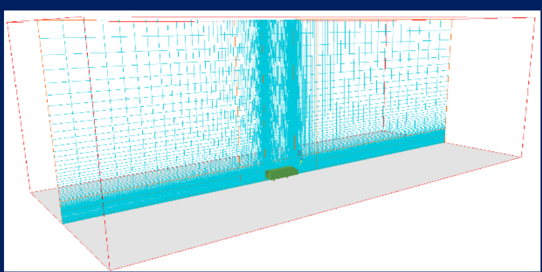
Turbulence is modelled using the Chen-Kim variant of K-epsilon formulation with a wall function treatment.

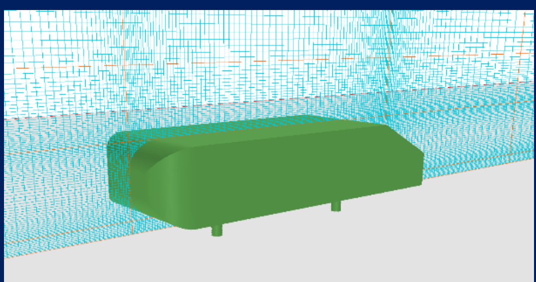
Meshing

Meshing is not an exact science, it needs some experimentation to get good results. With that being said, we aim to have fine detail in the areas near the vehicle and less detail further away.

Vehicular aerodynamics cases require a fine density of cells around the surface to be able to capture boundary layer behaviour in sufficient detail for accurate analysis. The areas in the leading and trailing edges of the vehicle are particularly important as stagnation and flow separation happen in those regions. To achieve this goal a mesh similar to the one below can be used – which had approximately 4millon cells.

It is good to remind ourselves that meshing can become a balancing act between grid refinement and simulation time, and therefore sometimes the user may have to decide which one to prioritise.





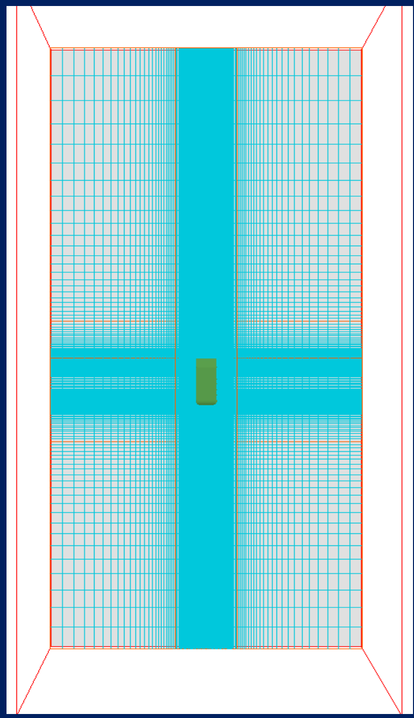


Figure 3 – grid used for this study

The grid shown above has the following number of cells most of which are concentrated in and around the vehicle:

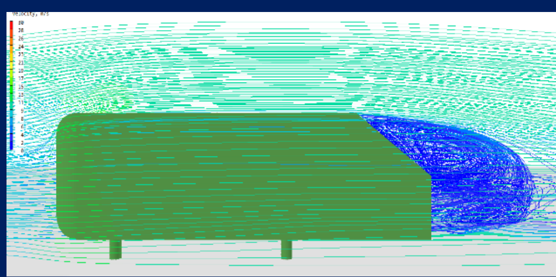
- 194 cells in the X axis
- 161 cells in the Y axis
- 119 cells in the Z axis
- 3.7 million cells in total

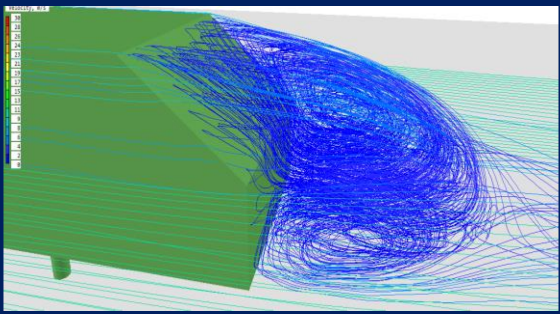
Results and discussion

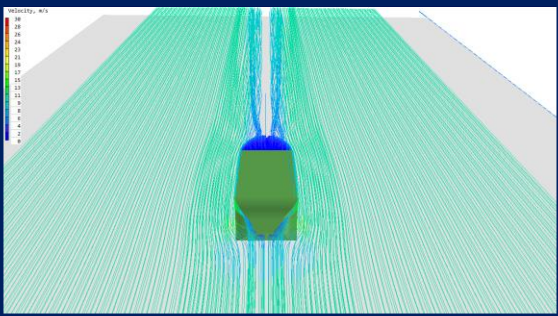
The goal of this study was to obtain the drag coefficient, for this the flow properties in the wake of the car need to solved with accuracy. The drag coefficient is defined as $C_d = {}^{2F_d}/_{\rho u^2 A}$, where F_d is the drag force computed from the simulation, ρ is the density, u is the free stream velocity and A is the projected frontal area without the struts.

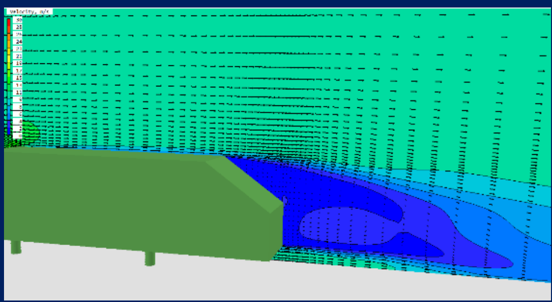
The images below show the flow structure within the wake region.

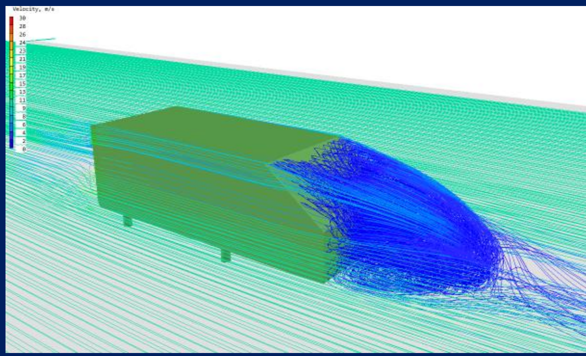
It is observed that a largely symmetrical recirculation region is formed in the wake, as would be expected from a steady state model.











The slant on the Ahmed used in our case study had an inclination of 25 degrees which from the experimental data corresponds to a Cd of 0.342 (Meile W. B., 2011).

In the tests carried out we obtained a computed Cd of 0.396. This result has an error of ~15% with experimental values when using a moderate grid resolution.

A re-run of the case with an approximately 20% finer grid resolution resulted in a Cd value of 0.333, this gave an error of ~3%. It can be hypothesised that further increases in grid resolution, eg: from a 4million case to 20-30million case would likely bring the error under 1%. However, computation time may become unfeasible at that point.

References:

W. Meile, et al, 2011. Experiments and numerical simulations on the aerodynamics of the Ahmed body. CFD Letter, 3(1), pp. 32-39.

Contact Us

CHAM provides software solutions, training, technical support and consulting services. Contact: Sales@cham.co.uk. For issues relating to PHOENICS Azure services contact: phoenics.cloud@cham.co.uk or call +44 (0)20 8947 7651.

Should you require any further information regarding our offered products or services please give us a call on +44 (20) 89477651. Alternatively, you can email us on sales@cham.co.uk.

Our website can be viewed at www.cham.co.uk and we are on the following social media:

Information regarding some of the new features to be found in PHOENICS-2022 can be found on page 2