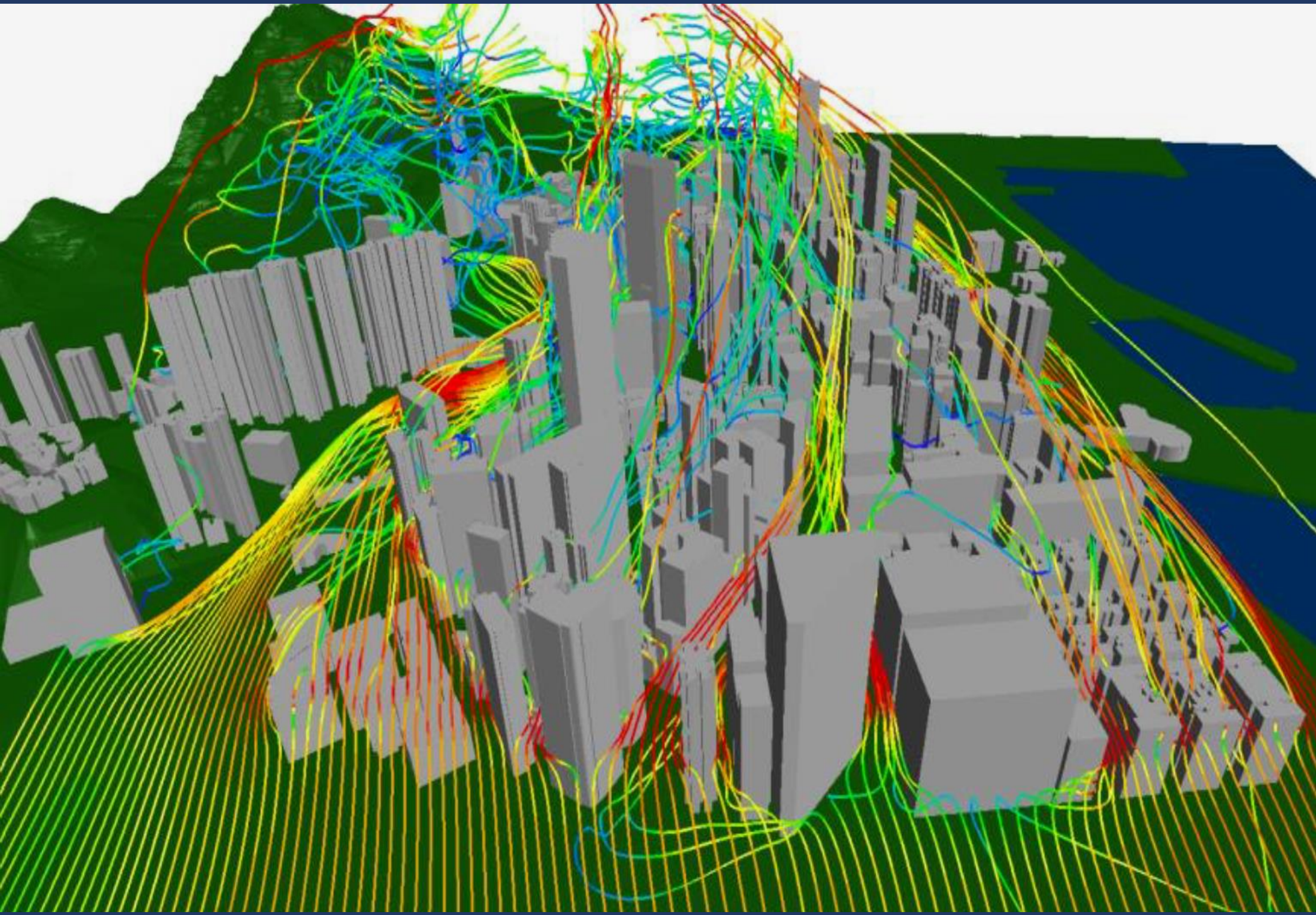
PHOENICS 2018

PHOENICS is the first, and the most established, commercial CFD Software available. It is reliable, cost effective, has a proven track record and can simulate scenarios involving fluid flow, heat and mass transfer, chemical reactions and combustion for a wide range of applications in industry and the built environment. PHOENICS 2018 is now available. Read further to see what new features have been added as well as what is planned for the next release. Contact us for more information.



CHAM

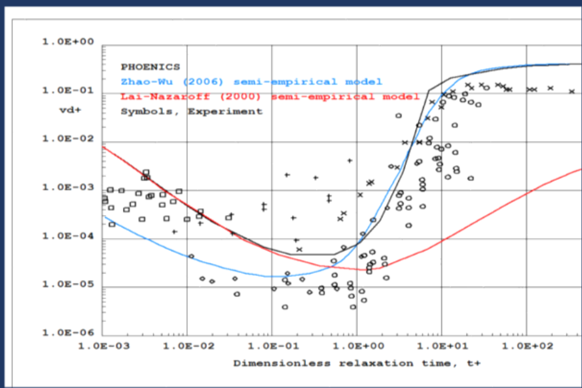
Experts in CFD Software and Consultancy

Drift Flux Model for Aerosol Deposition

An Eulerian-based multi-phase model for simulating the dispersion and deposition of aerosol particles in indoor environments is a standard option in PHOENICS-FLAIR 2018. Typical applications include studying indoor air quality and designing ventilation systems to deal with: human exposure to biological or radiological aerosols in healthcare or laboratory environments; health hazards from industrial aerosols; protective environments and isolated clean rooms; and surface contamination of artworks, electronic equipment, etc.

Aerosols can be deposited on surfaces by mechanisms including particle inertia, gravitational settling, Brownian diffusion (particles are transported towards the surface as a result of collisions with fluid molecules), turbulent diffusion (particles are transported towards the surface by turbulent flow eddies), turbophoresis (particles migrate down decreasing turbulence levels as a result of interactions between particle inertia and inhomogeneities in the turbulence field) and thermophoresis (temperature gradients drive particles towards or away from surfaces).

The model addresses these mechanisms apart from thermophoresis (to follow). Surface-deposition fluxes are calculated using semi-empirical wall models as a function of particle size, density and friction velocity. Deposition rates are reported automatically for all surfaces by the CFD solver.



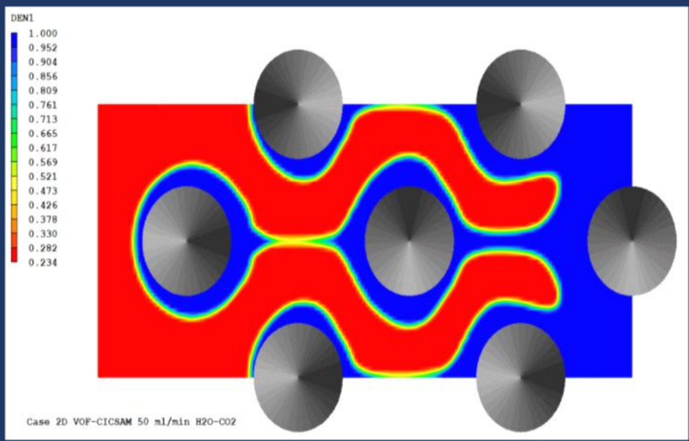
The new model has been validated successfully for particle deposition from fully-developed turbulent air streams in both horizontal and vertical ventilation ducts, and also from air moving in a laboratory-scale ventilation room. PHOENICS results agree well with measured data; and the “S-shaped" curve of deposition velocity versus particle relaxation time is well simulated as can be seen from the above diagram.

Free Surface Models

PHOENICS offers all Free-Surface / Volume of Fluid Models available in other CFD codes including:

- VOF-CICSAM (Compressive Interface Capturing Scheme for Arbitrary Meshes)
- VOF-HRIC (High Resolution Interface Capturing Scheme)
- VOF-MHRIC (Modified HRIC)
- VOF-STACS (Switching Technique for Advection and Capturing of Surfaces)

The above models offer a much sharper interface than the existing Height of Liquid (HOL) and Scalar Equation (SEM) Methods. All the models now available can include surface tension force, using the Continuum Surface Force Method.



Turbulence Models

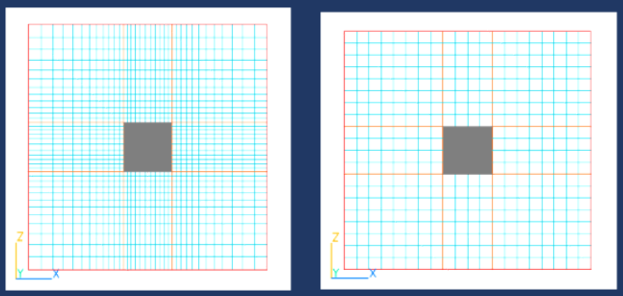
PHOENICS includes the following turbulence models as standard:

Revised Wilcox 2008 k-ω model: An improved version of Wilcox's (1988) model that incorporates the addition of a cross-diffusion term, a stress-limiter modification to the eddy viscosity; and a vortex-stretching modification to the ω equation.

Menter baseline 1992 k-ω model: Combining the k-ω and k-ε models to take advantage of the former's robustness near wall surfaces (due to its simple low Reynolds number formulation and its ability to compute flows with weak adverse pressure gradients accurately) and the latter's better performance near the boundary layer edge and away from walls, due to its insensitivity to free stream values. The technique is based on transforming the k-ω model to a k-ε formulation. This is an exact conversion, except for small contributions from the diffusion term due to the difference in the diffusion coefficients of the k and e equations.

Menter sst k-ω model: An extension of Menter's k-ω model that accounts for the transport of turbulent shear stress and so offers improved predictions of flow separation under adverse pressure gradients. The model differs from Menter's k-ω model in that a limiter is applied to the eddy-viscosity relationship.

Automeshing

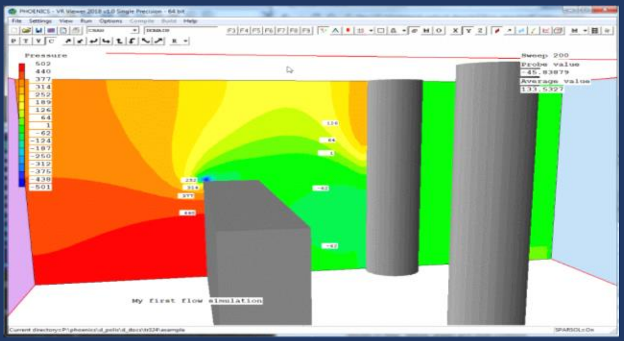


The image on the leftt shows the current authmesh and gives a comparison to an earlier version on the right.

Default Automesh settings mean that the behaviour of automesher is more predictable.The initial mesh setting in PHOENICS is always 1 cell per region. This is then refined until either the ratio of cell sizes across region boundaries satisfies the set criterion, or the smallest cell at a region boundary drops below the set minimum value. The resulting mesh is fine and should require less adjustment than previously. The automesher can be activated, by the user, to take into account cyclic boundaries

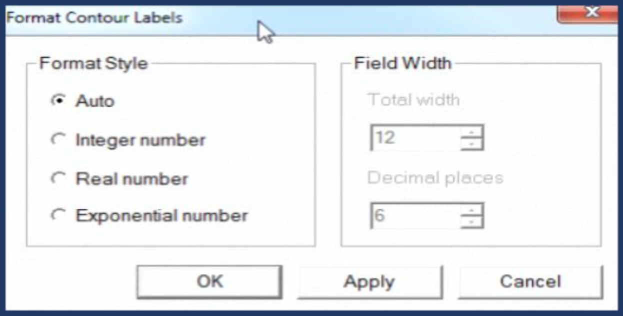
Contour Labelling

Lines separating contour bonds can be labelled automatically or manually.



If the manual option is chosen the format can be set to:

- Automatic, based on the size of numbers in the contour range
- Integer
- Real
- Exponential



For each of the three manual modes, the total field width and number of decimal places (if relevant) can be set.

Universal Thermal Climate Index

The Universal Thermal Climate Index UTCI provides an assessment of the outdoor thermal environment in bio-meteorological applications based on the equivalence of the dynamic physiological response predicted by a model of human thermoregulation, which is coupled with a state-of-the-art clothing model. The operational procedure (available from www.utci.org), shows plausible responses to the influence of humidity and heat radiation in the heat, as well as to wind speed in the cold and is in good agreement with the assessment of ergonomics standards concerned with the thermal environment.

In the FLAIR implementation, local air temperature is taken as solved temperature TEM1, local water-vapour pressure is derived from the solved water vapour mass fraction MH2O or a user-set constant, local mean radiant temperature is taken as the solved radiant temperature T3 or a user-set constant, and local wind speed is taken as local absolute velocity VABS.

Linked Angled-in Objects:

A pair of linked ANGLED-Ins can act as a filter if an additional source is set for scalars passing through a pair of linked ANGLED-IN objects. One can:

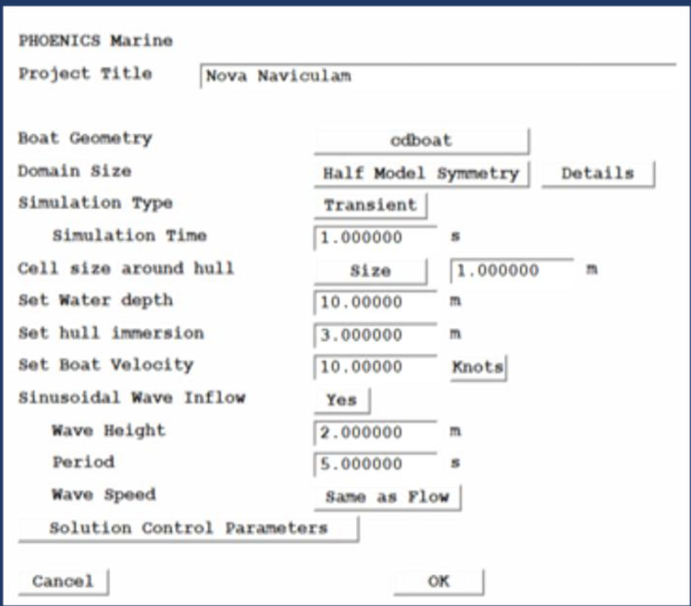
- Add a source (equivalent to adding a heat source to the energy equation);
- Add a fixed amount (equivalent to adding a temperature rise);
- Set the exit value to a fixed value; or
- Reduce the exit value by a set percentage.

If they act as the entrance to and exit from a duct and AGE is solved, the transit time through the duct can be added to the AGE at exit.

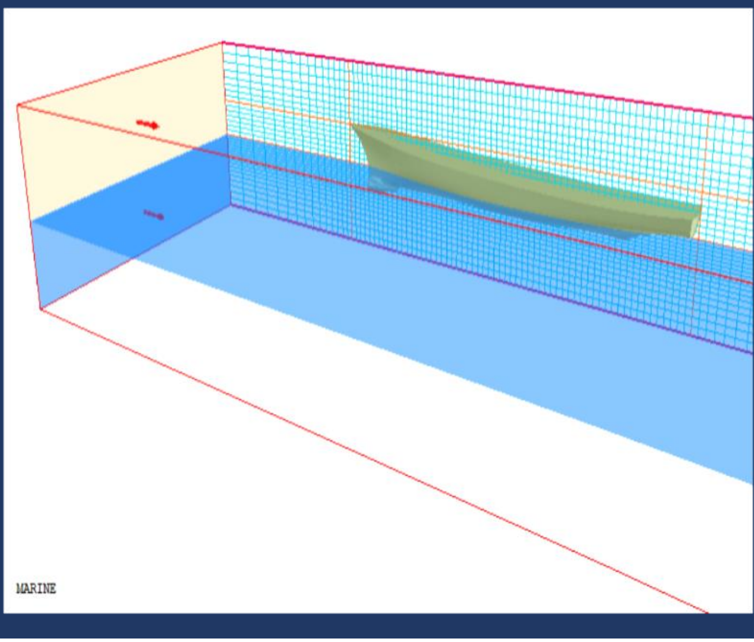
PHOENICS – Marine

CHAM is pleased to announce the availability of a new Special Purpose Product - PHOENICS-Marine, designed to enable Naval Architects and Marine Engineers to analyse hull performance quickly and effectively, and minimize the learning curve that CFD simulation methods sometimes present. A simplified menu interface, dedicated to this type of simulation, is used.

PHOENICS-Marine, employing free–surface options available in PHOENICS, assists users to import or specify particular hull shapes (and key parameters of simulations including flow velocity and waterline location) whilst automating aspects such as domain generation, mesh definition and selection of relaxation parameters.



PHOENICS-Marine will return values for drag, separated into skin friction and form drag, whilst displaying and quantifying the size of bow and stern waves produced. An added innovation allows users to test designs with sinusoidal waves as an input (see pg 11 of PHOENICS Newsletter Autumn 2017 for full description) and extract parameters such as pressure on the hull as a function of time. PHOENICS-Marine users will benefit from current developments which allow the hull to react dynamically to flow, enabling trim calculations of interest to naval designers. PHOENICS-Marine will also be activated in CHAM's Rhino-CFD, a Plug In to McNeel Corporation's Rhino3D CAD software. This should benefit naval engineers already using this package.



PHOENICS Development Roadmap 2018:

The current release of PHOENICS-2018 (v1.0 January 2018) is described in PHOENICS News, Winter 2017/208 and at: www.cham.co.uk/phoenics/d_polis/d_docs/tr006/tr006.pdf.

PHOENICS-2018- v2.0 – Summer 2018:

The next PHOENICS-2018 release is available mid-2018. New features in this version Include:

Auto wall functions for k- ω based models:

The present k- ω based models are divided into High- and Low-Reynolds Number groups which has an impact on grid spacing used for each group. The 'auto wall functions' remove this distinction by working out in each cell whether to use High- or Low-Reynolds models depending on distance to the wall and local velocity. Published methods will be implemented in PHOENICS. There is a possibility of applying something similar to k-e based models which would greatly improve the handling of near-wall grids using Parsol.

Full treatment of inclined surfaces for DFM:

An approximate treatment is currently available; full treatment will be implemented in the new release.

Marangoni effect and contact angle for all free – surface models:

This is included in the VOF-THINC feature with respect to the static contact angle. Consideration is being given to the dynamic contact angle where the contact angle is a function of local velocity.

Acceleration of parallel solver using HYPRE solver:

HYPRE is a library of high-performance solvers. The Moscow development team is working on re-coding parallel (and sequential) PHOENICS to make best use of the solvers it contains and optimise any advantages offered regarding equation solution. Preliminary test demonstrate that the solver AMG uses only some 20 iterations for the solution of a system with 10M cells for the pressure correction equation, but other solvers cannot provide the solution for an acceptable number of iterations (within 1000). A major innovation is that only 'live' cells are passed to the solver; this means that, for cases with many blockages, fewer cells need to be solved than is currently the case as 'dead' or fully-blocked cells will not enter the equation solver.

IPSA model improvements:

Work is underway to code in parts of the momentum equations to facilitate improvements in IPSA for modelling dense solid media (e.g. blast furnace, avalanche).

Thermo-phoresis for DFM:

Handling Thermo-phoresis, where temperature gradients cause particles to move along gradients will be implemented in the next release

Belgian / Dutch / FRS fire models (FLAIR)

These models will be available.

Cavitation model:

An InForm implementation of one validated cavitation model will be available via the GUI.

Shallow water model accessible through GUI:

This feature has been available through PIL for many years. In the new release it will be more easily accessible via the GUI.

Heat and scalar source at surfaces of facetted objects:

This is a feature which has been requested by users. It is intended to provide the functionality needed via an extension of the method used for angled-in / angled-out object.

VOF – THINC:

Coding is currently being tested and extra inputs provided in the GUI after which the feature will be merged into PHOENICS and available in the next release. Any users who wish to try this before release date should contact CHAM.

In addition to the above there are bug fixes and small enhancements to Core and FLAIR.

Future PHOENICS Releases:

Items planned for inclusion in next release of PHOENICS

- Inclusion of replacement multi-grid and coupled solvers
- Re-evaluation of Unstructured PHOENICS
- Continued improvement of FGV
- Revision of PARSOL (X-PARSOL)
- Curved 'thin plate' object.
- Flow-induced movement of blockages

Ongoing Development Items

Additional development resources have been allocated. The following tasks are being undertaken in parallel with the above items:

- Finalise Heatex Simscene with updated PHOENICS Direct.