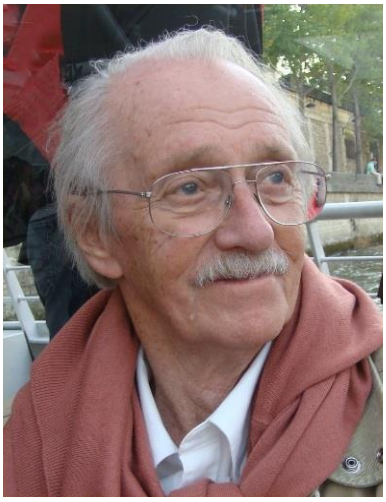
PHOENICS News

PHOENICS – Empowering Engineers   Winter 2017/2018

Professor D B Spalding FRS, FREng (1923–2016)

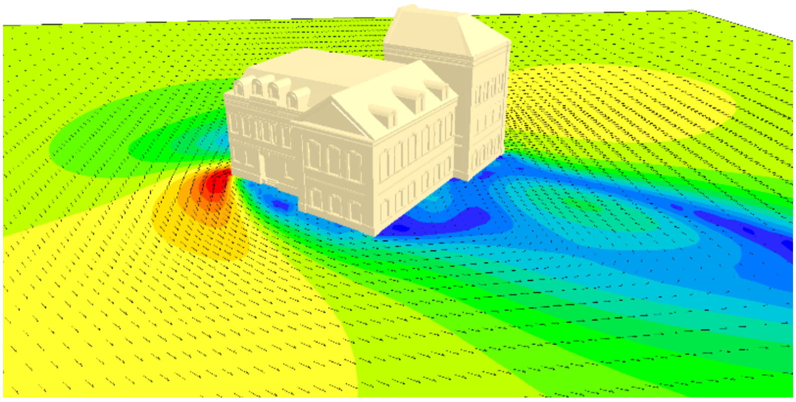
We invite Colleagues and Friends to attend a Celebration of the life and achievements of Professor Brian Spalding FRS, FREng at Imperial College London, on Friday April 20 2018. The event will start at 10:00 am and will comprise a series of short presentations covering various aspects of Brian's life as an engineer, scientist, person, poet and lover of life. If you would like to attend, and / or speak about Brian, please would you contact Professor Bill Jones who is arranging the event w.jones@imperial.ac.uk; copied to cik@cham.co.uk. Thank you. After the meeting there will be a dinner for those who wish to remain to which partners are also invited. The dinner cost will be around £50 per head. Further details can be obtained from Professor Bill Jones, email as above.



This Edition:

This newsletter describes new features in PHOENICS 2018 which is available, now, to new and maintained users. See pg 2 for further data. See pg 7 for an article from a PHOENICS user, describing the "*Modelling Pedestrian Wind Comfort in FLAIR and FLAIR-EFS.*"; Learn about the *"Prediction of Flame Compositions with a Discrete Reaction Model."*(pg 9), as well as News from CHAM Japan (pg 11).

If you would like to submit an article on PHOENICS, we would be delighted to hear from you. You can email us on news@cham.co.uk.
If you do not use PHOENICS and would like more information about the products and services we offer, please contact sales@CHAM.co.uk.



Wind velocities around mansion

PHOENICS Newsletter

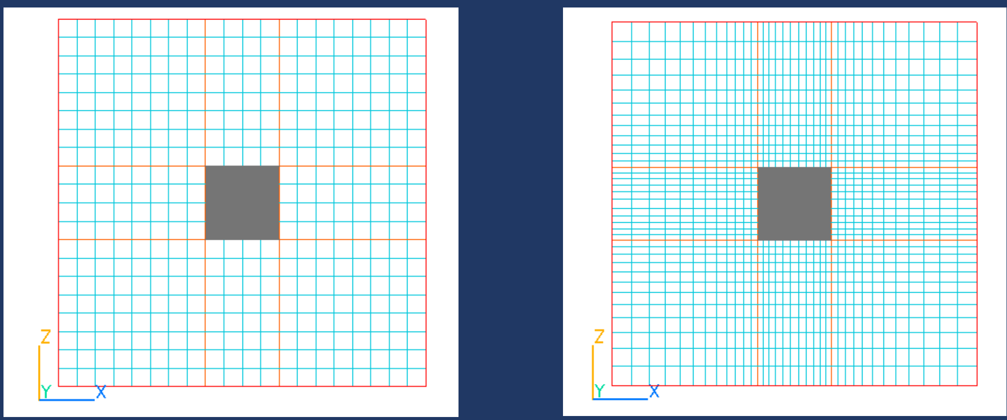
Contents: pg

What's New in PHOENICS 2018, by Mike Malin and John Ludwig, CHAM.

VR-Editor | Change to Automesh:

Default Automesh settings mean that the behaviour of the automesher is more predictable. The initial mesh is always 1 cell per region which is refined until either the ratio of cell sizes across all region boundaries satisfies the set criterion, or the smallest cell at a region boundary drops below the set minimum value.

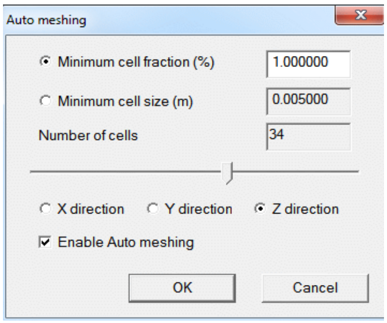
The resulting mesh is often finer than previously and should require less adjustment.



The image on the left shows the old Automesh, that on the right the new, both with default settings.

The automesher now takes into account cyclic boundaries activated by XCYCLE=T.

Interactive Mesh Change:



A right-click on the mesh brings up a dialog from which the mesh can be changed 'on the fly'. Moving the slider left decreases minimum cell size, so increasing cell number; conversely moving the slider right increases minimum cell size so decreasing total number of cells. For compatibility with existing cases, new rules are applied only to new cases, unless 'compatibility mode' is turned off on the Grid Mesh dialog for each direction. The new rules tend to give finer meshes than previously.

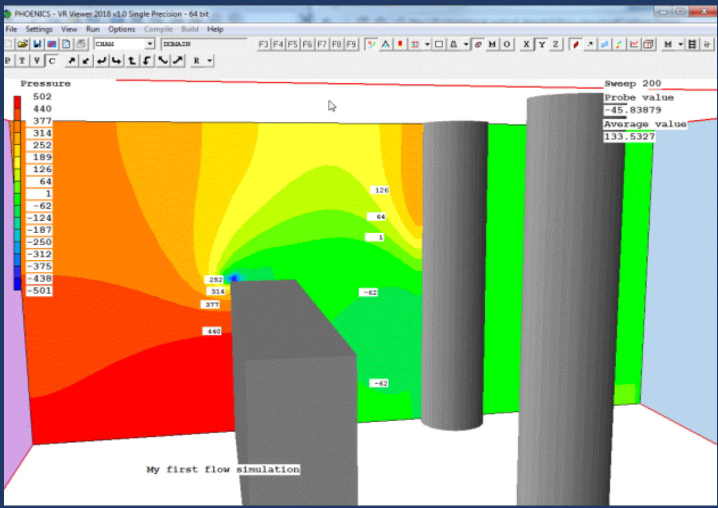
Linked Angled-in Objects:

An additional source for scalars passing through a pair of linked ANGLED-IN objects can be set:

- 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.

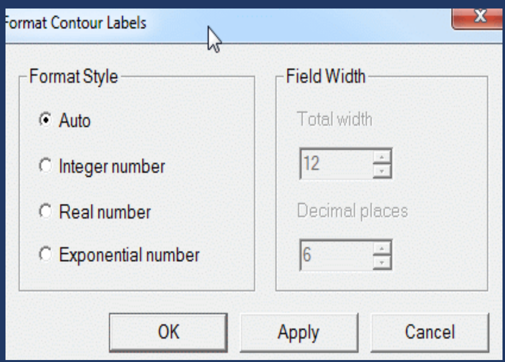
This allows a pair of linked ANGLED-INs to act as a filter. If they are acting as the entrance and exit from a duct and AGE is solved, the transit time through the duct can be added to the AGE at exit.

VR-Viewer - Labelling of Contours:



Lines separating the contour bands can be labelled automatically or manually.

The format used for contour labels and for the contour key can be set manually to:



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

For each manual mode (3), total field width and number of decimal places (if relevant) can be set.

Q1 – PIL:

New PIL variable **ISCHM** – denotes first sweep at which higher-order differencing schemes are active. The default setting is 1 for compatibility with existing cases. Setting it greater than 1 allows the first-order solution to develop before activating the higher-order scheme.

New PIL variable **ISURFA**. This, in conjunction with the existing HOL and SURF flags, controls which free-surface model to use. When SURF=T and HOL=F, ISURFA means:

- 0 – use SEM
- 1 – use VOF-CICSAM
- 2 – use VOF-HRIC
- 3 – use VOF-MHRIC
- 4 – use VOF-STACS

Existing SEM case can be converted to VOF by adding ISURFA=n to Q1 where n chooses the VOF model.

New PIL variable **SURFTA**. This sets the surface tension constant.

Solver – Earth

New and Upgraded Turbulence Models:

The following turbulence models have been added:

- The Revised Wilcox 2008 k-ω model is an improved version of Wilcox's (1988) model that retains the strengths of the original model for wall-bounded flows, but improves the model's predictions for free shear flows, as well as significantly reducing the model's sensitivity to the free-stream values of ω. These improvements are achieved by the addition of a cross-diffusion term, and a vortex-stretching modification to the ω equation that resolves the round-jet anomaly. In addition, a stress-limiter is applied to the eddy viscosity, which improves the model's accuracy for separated flows by accounting for shear-stress transport.
- The Menter Baseline 1992 k-ω model, which combines the k-ω and k-ε models by using a blending function to switch gradually from the k-ω model in the near-wall region to the k-ε model near the boundary-layer edge. This means that the k-ω model's superior near-wall low-Reynolds-number formulation is retained with its advantages of better accuracy and numerical stability; whilst towards the boundary-layer edge and away from walls, the k-ε formulation is recovered with its insensitivity to free-stream values.
- The Menter SST (Shear Stress Transport) k-ω model extends the Baseline k-ω model to account for the transport of turbulent shear stress by applying a limiter to the eddy viscosity. This extension offers improved predictions of flow separation under adverse pressure gradients, and the model also includes a production limiter to prevent excessive turbulence production in stagnation regions. The model is known to perform well for a wide range of applications, but it often requires good resolution of the boundary layer for accurate predictions, and it can be more difficult to converge than the standard k-ω and k-ε models because it is more non-linear.

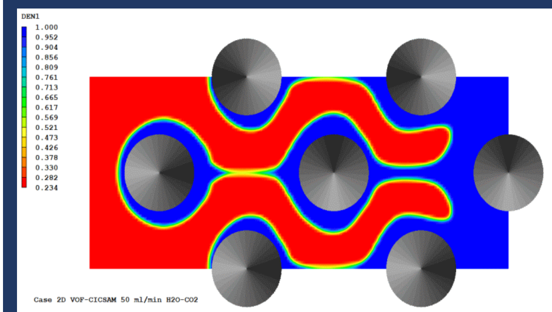
The above three k-ω models are made available in high- and low-Reynolds Number form, and can be activated through the VR Editor interface, or by the TURMOD command in Q1. The constants for all the turbulence models can be reset from Q1 using SPEDAT commands, and the values used and model name are echoed at the top of RESULT.

New Free Surface Models:

The following Volume-of-Fluid models have been added:

- 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 choice of these methods depends on the Courant number value. For Courant numbers smaller than 0.3 CICSAM should be chosen. For values of Courant numbers below 0.5 and above 0.3, the HRIC and modified HRIC can be used. For higher values of Courant number, the STACS and MHRIC should be preferred. For large domains (like flow around ships), HRIC and MHRIC are preferable. The above models, as well as the existing HOL and SEM methods, can include surface and tension using the Continuum Surface Force Method.



Near Critical CO_2 Flowing in a Channel with Octagonal Micro-pillars filled with water.

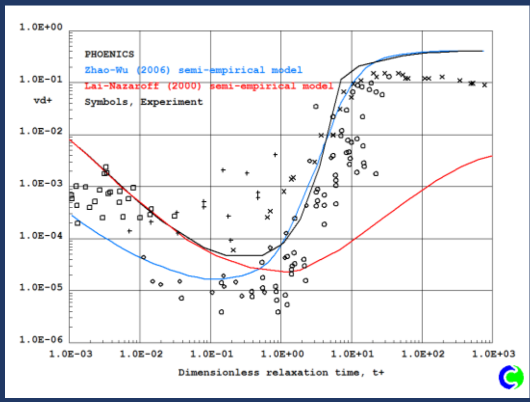
FLAIR – Drift Flux Model for Aerosol Deposition:

An Eulerian-based multi-phase model for simulating dispersion and deposition of aerosol particles in indoor environments has been implemented as 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.

The aerosol model assumes a very dilute particle phase (one-way coupling) with no collisions or coalescence, and drift-flux modelling is used to represent slippage between the particle and gas phases due to gravitational effects. In practice, aerosols can be deposited on surfaces by various mechanisms, including particle inertia, gravitational settling, Brownian diffusion (where particles are transported towards the surface as a result of their collision with fluid molecules), turbulent diffusion (where particles are transported towards the surface by turbulent flow eddies), turbophoresis (where particles migrate down decreasing turbulence levels as a result of interactions between particle inertia and inhomogeneities in the turbulence field) and thermophoresis (where temperature gradients drive particles towards or away from surfaces).

The PHOENICS model considers all these mechanisms apart from thermophoresis, which is planned for a future release. The surface-deposition fluxes themselves are calculated by using semi-empirical wall models as a function of particle size, density and friction velocity, and the deposition rates are reported automatically for all surfaces by the CFD solver. There are four alternative deposition models, and these include a formulation which accounts for any distance from the wall, rather than assume that the near-wall grid point lies outside the particle concentration boundary layer in fully-turbulent flow. At present this particular deposition model doesn't account for turbophoresis, but it is especially useful for cases where the near-wall grid point lies in the laminar or transitional region of the boundary layer.

The new aerosol 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. For vertical ducts, inertial impaction and gravitational settling are absent, so this case provides a test of deposition influenced by molecular and turbulent processes. The PHOENICS results agree well with the measured data, and the "S-shaped" curve of deposition velocity versus particle relaxation time is well simulated, as can be seen from the diagram in the next column.



Universal Thermal Climate Index

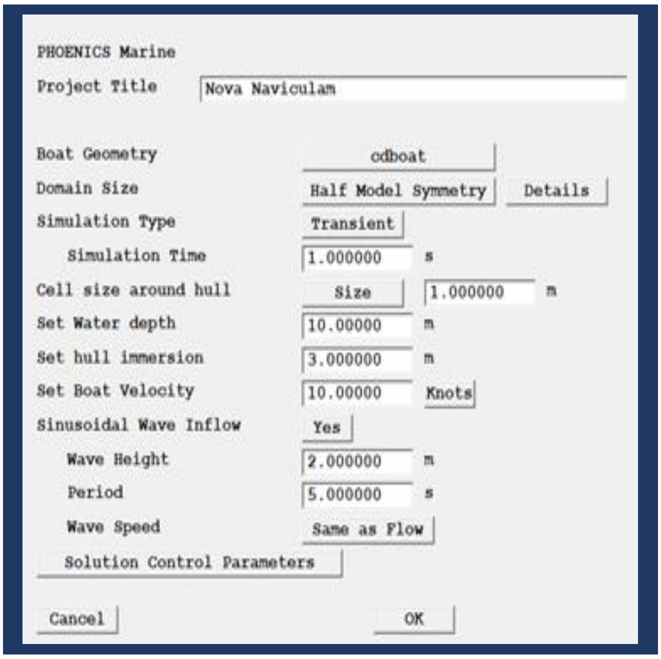
The Universal Thermal Climate Index, UTCI, provides an assessment of the outdoor thermal environment in bio-meteorological applications based on the equivalence of the dynamic physiological response predicted by a model of human thermoregulation, which is coupled with a state-of-the-art clothing model. The operational procedure, which is available as software from the UTCI website (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.

The necessary research for this was conducted within the framework of a special commission of the International Society of Biometeorology (ISB) and European COST Action 730.

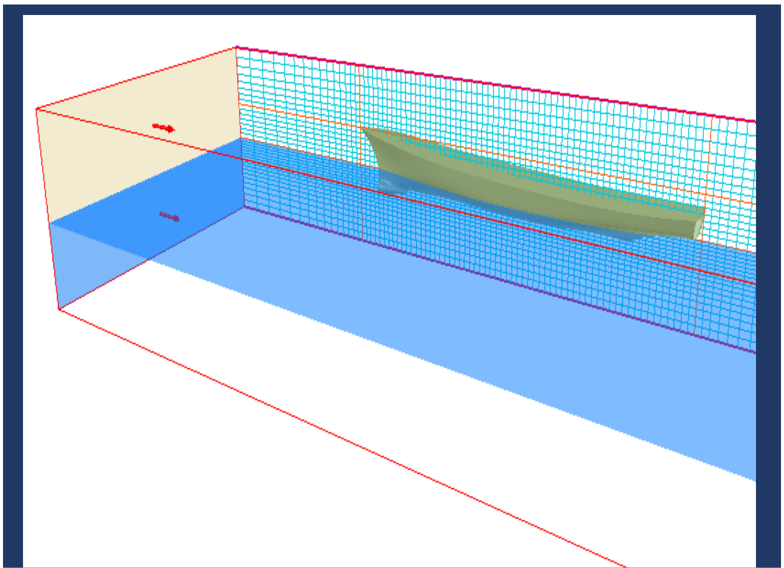
The published subroutine returns the UTCI as a function of air temperature, water - vapour pressure, mean radiant temperature and wind speed 10m above ground level. In the FLAIR implementation, the local air temperature is taken to be the solved temperature TEM1, the local water - vapour pressure is derived from the solved water vapour mass fraction MH2O or from a user-set constant, the local mean - radiant temperature is taken as the solved radiant temperature T3 or a user-set constant, and the local wind speed is taken to be the local absolute velocity VABS.

PHOENICS-Marine:

CHAM is pleased to announce the availability of a new Special Purpose Product (SPP), called 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 their particular hull shape (and key parameters of their simulations including flow velocity and waterline location) whilst automating aspects such as domain generation, mesh definition and selection of relaxation parameters.



PHOENICS Marine

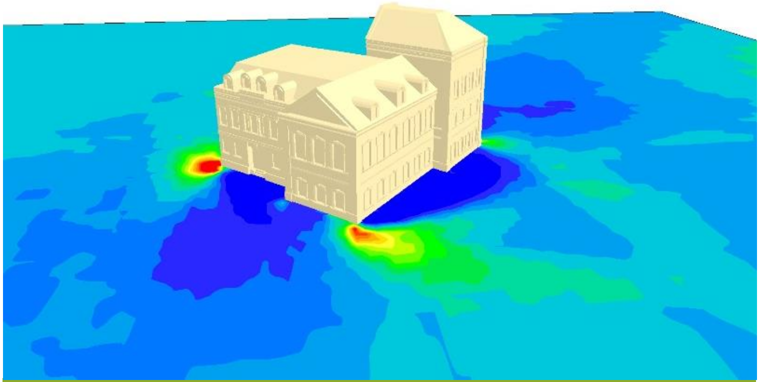
PHOENICS Marine will return values for drag, separated into skin friction and form drag, while displaying and quantifying size of bow and stern waves produced. An added innovation allows users to test their designs with sinusoidal waves as an input - see pg 11 of PHOENICS Newsletter Autumn 2017 for full 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 the trim calculations of interest to naval designers. PHOENICS Marine will also be activated in CHAM's CFD "Plug In" to McNeel Corporation's Rhino3d CAD software, and should benefit greatly the large number of naval engineers who already use this package.

Modelling Pedestrian Wind Comfort in FLAIR and FLAIR-EFS

By David Glynn, CHAM Consultant:

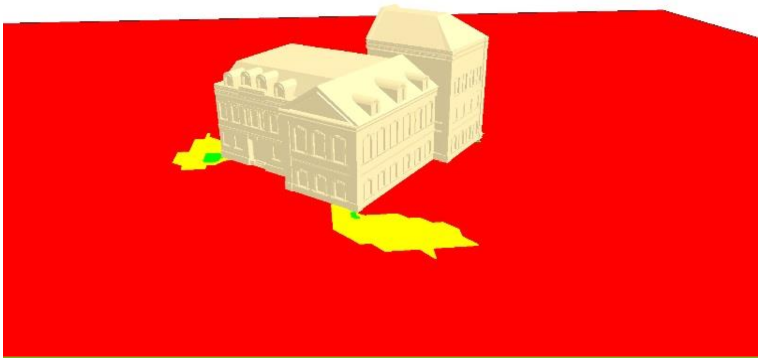
When new buildings are constructed, particularly high-rise buildings commonly seen in modern developments, wind speeds in the environs of the buildings are affected. This can lead to uncomfortable or even dangerous conditions for pedestrians. Many city authorities will grant permission for new high-rise developments only if a wind comfort study indicates that the consequences for the pedestrian environment will be satisfactory. Such studies often involve CFD simulations which make use of statistical meteorological data, and will typically require running cases with wind directions from various points of the compass. The results of these simulations need to be post-processed to generate the required statistical wind-comfort parameters, of which the following are now available in FLAIR: (1) average velocity over wind speeds and sectors, (2) probability of exceeding a threshold velocity, (3) Lawson criteria, (4) NEN 8100 categorisation. These are explained below.

Wind-frequency data are input into FLAIR or FLAIR-EFS in the form of a "Wind Data file", typically based on wind data obtained over a period of several years. The file specifies the probability of the wind having a particular speed and a particular direction. More specifically - the total range of possible wind speeds is divided into a specified number of intervals, and the 360° range of possible wind directions is divided into a specified number of sectors. For every speed interval and every sector, a probability value is provided.



Probability of wind exceeding 6 m/s - red indicates 10%

A new and much improved procedure is now available in FLAIR and FLAIR-EFS for generating wind-comfort parameters. To activate the wind averaging, simply set "Store average velocity over all sectors" to ON in the Comfort Indices Settings panel. In the same panel, specify the name of the Wind Data file containing the frequency data. In Wind Object Settings set the wind direction as usual, but do NOT set wind speed; this is defined automatically to be the weighted average for the given direction over the range of wind speeds, based on the probabilities in the data table.



Lawson categories - red indicates (5), yellow (4), green (3)

After the last run has been made, the output files of all the runs must be passed through a utility program, PHISUM, which does the necessary averaging and produces statistical data based on the given wind frequencies.

PHISUM is activated from "Run" / "Utilities" on the top bar of the VR Editor, and asks the user to specify the ".pda" file names containing the results of the simulations for each sector modelled. The values of average velocity over the wind speeds and sectors are stored in variable VAV in the output file from PHISUM. If this file is attached to the Viewer, together with one of the Q1 files from the individual runs, then VAV can be plotted in the usual way.

The required statistical parameters can be plotted from the output file of PHISUM, to generate various assessments of pedestrian comfort, as follows:

1. “Average velocity over wind speeds and sectors” - variable name VAV.
2. “Probability of exceeding” - variable name PRO. This is the probability, between 0 and 1, of the wind speed at each individual cell exceeding a specified threshold value.
3. “Lawson Criteria” - variable name LAWS. The Lawson Comfort Criteria define a range of pedestrian activities: (1) roads and car parks, (2) fast walking, (3) pedestrian walk-through, (4) pedestrian standing, (5) sitting. For each activity a wind speed and maximum frequency of exceedance are defined. If the probability of the wind speed exceeding this threshold exceeds the given limit for the activity, then conditions are deemed to be unacceptable. The above categories are ordered, from the least comfortable to the most comfortable. LAWS indicates the highest-number activity for which the conditions will be acceptable.

Thus, for example, if at a certain place in the domain condition (5) was met, then conditions would be deemed comfortable for sitting, and LAWS would be set to 5. But, in a more consistently windy location only condition (1) might be met, in which case LAWS would be set to 1, meaning that so far as pedestrian comfort is concerned this area is suitable only for roads or car parks. Plots showing the areas in each Lawson category are frequently found in CFD assessment reports for proposed building developments.

4. “NEN8100” - variable name NEN. This is a Dutch standard for wind comfort and wind danger in the built environment, similar in character to Lawson but different in detail. Five comfort ranges and two danger ranges are defined. The variable NEN indicates which range is applicable for each pedestrian area.

The pictures illustrate the models for the case of wind around a historic mansion. The first picture shows the probability of the local wind (from any direction) exceeding 6 m/s. The spots which are likely to be the windiest are close to the corners of the mansion. The second picture shows a plot of the Lawson categories. Green is comfortable for pedestrian walk-through only; yellow for standing; and red for sitting. So if, for example, one was wanting to choose a location for outdoor seating for the cafeteria, one would not choose the yellow areas near the corners of the mansion.

For a fuller explanation of the above, see the section on Comfort Indices in the FLAIR-EFS User Guide TR316, together with the Appendix of that document which gives details of the Wind Data file (where the wind frequency data are specified).

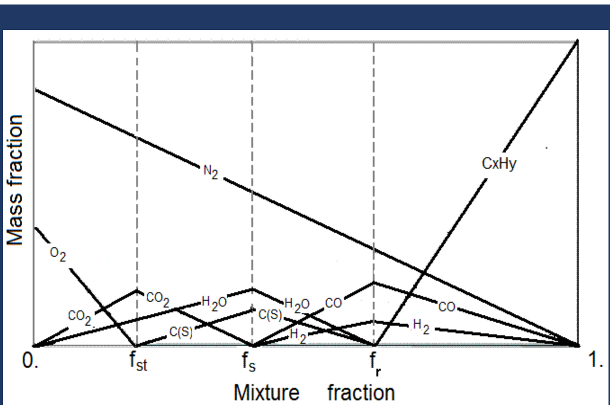
Prediction of Flame Compositions with a Discrete Reaction Model:

By S.V. Zhubrin | Independent Researcher | January 2018 | svzhubrin@yahoo.co.uk

This PHOENICS development provides Combustion Engineers with a reasonably simple, yet realistic and comprehensive method, by which they may mathematically predict the complex composition of industrial flames. With this in mind, a group of global chemical reactions for combustion of hydrocarbon fuels, originating from discretization of oxidation chemistry, has been developed and implemented in PHOENICS.

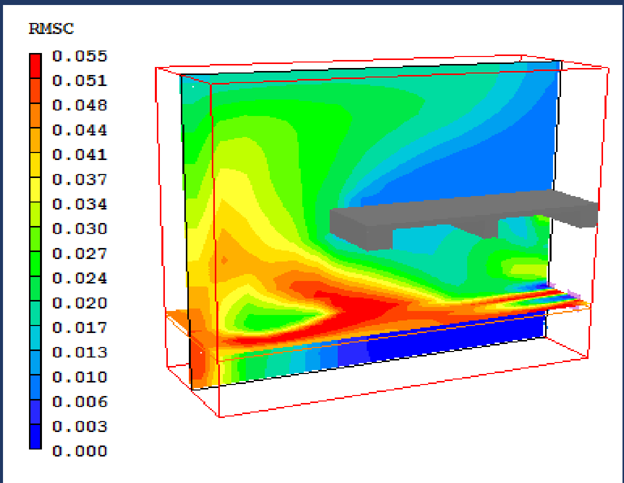
The key concept stems from the computational analysis of flame compositions from a viewpoint of considering chemical elements, which are conserved through chemical reactions, as well as the species mass fractions of the gas mixture [1-2]. An elementary theory of Simple Chemical Reaction Scheme, (SCRS), proposed elsewhere [3], has been recovered by the author from consideration of the balances in five-gas chemical reactions of complete fuel oxidation when discretised with respect to either the oxygen-element or mixture-fraction contents [4]. On this basis, the general principle underlying the discrete reaction analysis has been worked out. It is that the assumption of discretized representation of oxidation chemistry allows, by restrictions on the number of product species involved, and the postulation of fast reaction rates, the deduction of a closed set of algebraic equations, and leads to their analytical solutions in terms of the mass fraction of product species.

The methodology of reaction discretisation is defined as DRM, Discrete Reaction Model. Based on this model a number of advanced discrete reactions have been elaborated, for which the mass fractions of main combustion intermediates are shown to be linked to the elemental mass composition with associated flame-limiting requirements [4, 5]. The types of global mechanisms developed include intermediate, conversion, rich, and sooting five-species reactions; six-species incomplete oxidations with the presence of carbon monoxide, un-combined hydrogen and carbon soot; and seven - gases compositions (with and without soot). The results of the tests, validation exercises and application case studies suggest that DRM is, generally, in reasonable agreement with the wide range of experimental observations, and shows correct trends [5-9].



Mixing and flame-limit discretization of reactions between fuel and oxidant streams producing soot, hydrogen and carbon monoxide intermediates [5]

The DRM framework is flexible enough to accommodate the models of turbulence chemistry interactions, as was done in [7] the Probability-Density-Functions method [7], and in [8,9] with the Stream Recognition Model of transported probabilities. Model formulations, being of an algebraic nature, are easy to import into the one- or multi-phase solvers of available CFD software. Reaction discretisation has proved to be a powerful technique for predicting complex composition of reacting flows of combustible mixtures in a variety of engineering applications. Full details of the DRM equations, validation case studies and selected engineering applications are publicly available in [4-9].



Contours for RMS of CO mass fraction fluctuations at central plane of a secondary combustion chamber predicted by Stream Recognition Model of discrete turbulence-chemistry interactions [8]

References:

1. D.B. Spalding, A standard formulation of the steady convective mass transfer problem, Int. J. Heat Mass Transfer 1 (1960) 192–207.
2. D. B. Spalding, Convective Mass Transfer; an Introduction, Edward Arnold, London, (1963), pp. 384.
3. D. B. Spalding, Combustion and Mass Transfer, Pergamon Press, London, (1979), pp. 410.
4. S. V. Zhubrin, Development of Discrete Reaction Model for Turbulent Combustion: Discretely Reacting systems, October (2010), Available at https://dx.doi.org/10.13140/2.1.4452.6880
5. S. V. Zhubrin, Discrete Reaction Model for Composition of Sooting Flames, International Journal of Heat and Mass Transfer 52(17):4125-4133, August (2009), Available at https://dx.doi.org/10.1016/j.ijheatmasstransfer.2009.03.034
6. S. V. Zhubrin, Prediction of Three-Stream Oxidation in a Secondary Combustion Chamber with Discrete Reaction Model, July (2009), Available at https://dx.doi.org/10.13140/RG.2.1.1906.3846
7. Xu B.P. Chen Z.B., J.X. Wen, S.V. Zhubrin, S. Dembele, Large Eddy Simulation of a One-Metre Diameter Methane Fire using the Extended Discrete Reaction Model, January (2011), 6th International Seminar on Fire and Explosion Hazards, Available at https://dx.doi.org/10.13140/2.1.2851.1523
8. S. V. Zhubrin, Prediction of Fluctuating Properties in Discrete Reaction Model for Turbulent Combustion, May, (2011), Available at https://dx.doi.org/10.13140/RG.2.1.4090.0004
9. S. V. Zhubrin, Computations of Probabilistic Chemistry-Turbulence Interactions by Stream Recognition Model of Premixed Combustion, July (2017), Available at https://dx.doi.org/10.13140/RG.2.2.27744.79367

News from CHAM – Japan

Content: Photographs, and contributions titles from CHAM Japan's PHOENICS User Day held in October 2017.

By Jeremy Wu

小風量熱源の準恒温槽への適用確認と評価
吉野石膏株式会社　竹中　彪
Confirmation and evaluation of application of small air flow heat source to quasi-constant temperature bath.
Yoshino Gypsum Co.,Ltd, TAKENAKA TAKESHI

船舶艤装設計における流体解析の PHOENICS による活用事例
九州大学　海洋システム工学部門　篠田　岳思
Example of utilization of fluid analysis by PHOENICS in marine outfit design.
Kyushu University, SHINODA TAKESHI

PHOENICS を用いた伝熱計算の検討と評価
積水化学工業株式会社　水野　絢可
Study and evaluation of heat transfer calculation using PHOENICS.
Sekisui Chemical Co., LTD., MIZUNO AYAKA

要素モデル内水ー蒸気２相流解析を基にした地熱貯留層評価
熊本大学　地圏環境エネルギー分野　佐藤　晃
Evaluation of geothermal reservoir based on water-vapor two-phase flow analysis in element model
Kumamoto University, SATOH AKIRA

風が流れるかたち
株式会社石元建築事務所　菅原　雄一郎
The way the wind flows.
Ishimoto Architectural & Engineering Firm, Inc., SUGAWARA YUICHIRO

スイマーにかかる前泳者のバタ足の影響
龍谷大学　機械システム工学科　塩見　洋一
The influence of the legs of the swimmer in the front.
Ryukoku University, SHIOMI YOICHI

Inform を使用した計算サンプル
鈴木　俊之
Some simulation samples using Inform
CHAM Japan, SUZUKI TOSHIYUKI

News from CHAM

VACANCIES:

If you seek an interesting career in a small software house specialising in Computational Fluid Dynamics (CFD) CHAM has two vacancies at its Head Office in Wimbledon. We have opportunities, at junior level, for:

1) CAD Engineer with solid skills in C++, C# .Net framework, and; ideally, Grasshopper to work on CAD and interface development with PHOENICS and subset products such as RhinoCFD (integrating CFD capabilities into Rhino3D). The ideal candidate will have spent time using and developing software/plug-ins and either have experience in the commercial world or have been exposed to commercial expectations during time at University.

2) Development Engineer to assist the Software Development Team develop, implement, test and document new features, or modifications of, PHOENICS and other CHAM products. The aim is to create viable, and saleable products within an agreed time frame. The successful candidate will also work with the User Support Team to investigate, find, report and assist with the solution of problems raised by CHAM's User Community.

The positions require a BSc or above in Computational Science, Computer Aided Design, Systems, Mechanical or Aeronautical Engineering or related fields and a knowledge of CFD. The successful candidates, in addition to their technical qualifications, will possess excellent communication skills (verbal and written), enjoy working in a collaborative team environment, and know when to work independently and when to seek advice.

PHOENICS/FLAIR Training Course |09:30 -17: 00; 16-18 April 2018 | The Cityview Hotel, Kowloon, Hong Kong:

We are pleased to announce that CHAM's Senior Technical Support Consultant, Dr David Glynn, will present a 3-day training course in the use of its PHOENICS/FLAIR CFD software. The programme will introduce delegates to PHOENICS' interactive graphical environment for the simulation of fluid flow and heat transfer processes. Topics covered will focus on modelling internal and external flows occurring in the built environment including HVAC, thermal comfort, fire/smoke/pollution hazards, urban flow modelling and heat isles. Participants do not require prior experience in CFD simulation. They will have the opportunity to view and use the software during hands-on workshop sessions, and benefit from post-course familiarisation via an inclusive two-month, fully-supported licence of PHOENICS/FLAIR & FLAIR-EFS. Course content will be of interest to prospective and current PHOENICS users. The programme is based on PHOENICS 2018 and highlights some of its newer features including:

- New turbulence models, including Realisable k-e, k-w- SST, etc
- Scalable wall functions
- Lawson criteria for wind comfort
- Improved rain model
- Air Exchange Effectiveness for rooms
- Age of air for individual rooms
- Labelling of contour lines in Viewer
- Drift-Flux Model (DFM) for particle deposition & transport
- Universal Thermal Climate Index (UTCI) Comfort Index
- Inclusion of Belgian / Dutch / FRS fire models
- New rapid auto-mesh grid generation

The event is arranged in cooperation with the Meinhardt Group and Advanced Technovation Ltd. For further information, course fees and booking, please contact: Advanced Technovation Limited., G.P.O Box 11935, Hong Kong Contact: Albert T. Yeung, Phone: +852 91208024, E-mail: support@adv-technovation.com

Staff

We would like to welcome Robynne Moynihan who has joined us as a Marketing and Sales Promotion Administrator at CHAM.

Contact Us

Should you require any further information on any of our offered products or services, please give us a call on +44 (20) 8947 7651. Alternatively, you can email us on sales@CHAM.co.uk

Our website can be viewed at www.CHAM.co.uk

Concentration Heat and Momentum Limited

Bakery House
40 High Street
Wimbledon Village
London SW19 5AU,
England.