

PHOENICS – Your Gateway to CFD Success

Documentation for PHOENICS | TR 006

What's New in PHOENICS (Older Versions):

Updates and Additions:

CHAM Ref: CHAM/TR006

Document rev: 3

Doc. release date: 14 July 2022

Software version: PHOENICS 2015-19

Responsible author: J C Ludwig

Other contributors: M R Malin

Editor: J C Ludwig, R Moynihan

Published by: CHAM

Confidentiality:

Classification: Unclassified

The copyright covers the exclusive rights to reproduction and distribution including reprints, photographic reproductions, microform or any other reproductions of similar nature, and translations. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, electrostatic, magnetic tape, mechanical, photocopying, recording or otherwise, without permission in writing from the copyright holder.

©Copyright Concentration, Heat and Momentum Limited 2022

CHAM, Bakery House, 40 High Street, Wimbledon, London SW19 5AU, UK

Telephone: 020 8947 7651 Fax: 020 8879 3497

E-mail: phoenics@cham.co.uk, Web site: <https://www.cham.co.uk>



What's New in PHOENICS (Older Versions)

Contents

Items in PHOENICS 2019 v1.1 (July 2019)	1
VR-Editor	1
VR-Viewer	1
Solver- Earth	1
Items in PHOENICS 2019 v1.0 (March 2019)	4
VR-Editor	4
Flair / Flair EFS	6
VR-Viewer	6
Solver	9
CVD	14
PIL	14
Generic	14
Items in PHOENICS 2018 v1.3 (November 2018)	16
VR-Editor	16
Flair / Flair-EFS	16
VR-Viewer	16
Solver	16
CVD	16
Generic	16
Items in PHOENICS 2018 v1.2 (July 2018)	17
VR Editor	17
VR-Viewer	17
Solver	17
Flair-EFS	17
Generic	17
Items in PHOENICS 2018 v1.1 (March 2018)	18
VR-Editor	18
Solver – Earth	18
Items in PHOENICS 2018 v1.0 Added since the last release of PHOENICS 2017 (dated March 2017)	19
VR-Editor	19
VR-Viewer	20
Q1 – PIL	21
Solver – Earth	22
FLAIR	24
PHOENICS-Marine	25
Directory Structure	26
Items in PHOENICS 2017 added since the last release of PHOENICS 2015 (dated October 2016)	27
VR-Editor	27

Q1 – PIL	29
VR-Viewer	29
Solver – Earth	29
Flair / Flair-EFS	32
ESTER	33
Directory Structure	33
Items in PHOENICS 2015 added since the last release of PHOENICS 2015 (dated 31 March 2016)	34
VR-Editor	34
Q1 – PIL	35
VR-Viewer	35
Solver – Earth	35
Flair / Flair-EFS	37
Items in PHOENICS 2015 added since the last release of PHOENICS 2015 (dated 26 Oct 2015)	39
VR-Editor	39
VR-Viewer	39
Solver – Earth	40
Flair / Flair-EFS	41
Items in PHOENICS 2015 added since the last release of PHOENICS 2014 (dated 19 Jan 2015)	43
VR-Editor	43
Q1 - PIL	44
VR-Viewer	44
Solver - Earth	46
Flair-EFS	47
Flair	48
ESTER	48
CVD	48
General, VRE & VRV	49

For information about even earlier versions of PHOENICS, see 'The PHOENICS Chronicle', available on-line at http://www.cham.co.uk/phoenics/d_polis/d_chron/chron.htm

Items in PHOENICS 2019 v1.1 (July 2019)

The new items in this version are mainly corrections for problems reported in the previous release.

VR-Editor

General

The 'Edit InForm 13' button on the USER_DEFINED and CELLTYPE object dialogs has been restored at the request of users.

When a scalar source was set on a non-cuboid non-participating (material 198) blockage, the source setting was not echoed to the Q1 file. This omission is corrected.

When the name of the restart file was changed interactively, if the old name and new name both included a drive letter (e.g. C:), the change would not be accepted. This is fixed. The correct name could always be edited into the Q1 file.

For combustion cases, the option to set the external gas composition to 'in-cell' (equivalent to PIL SAME) has been added to fixed pressure boundaries (OUTLET, ANGLED-OUT).

The ability to set maximum sweep-to-sweep increments when CONWIZ=F has been revised using the CONWIZ mechanism. The previous method of implementing this was obscure and resulted in the loss of being able to set minimum and maximum values.

Flair

When the Drift-Flux Model is active, the WIND object now automatically creates the appropriate deposition settings on the ground plane.

Ester

When output tables of currents at the collector bars and potentials at the anodes are requested, the tables are restricted to 10 columns per file. If more than 10 columns are needed, these will be written to separate files. This avoids internal problems when there are too many columns per file.

VR-Viewer

The direction of individual streamlines can be switched between upstream, downstream or both after they have been generated. Previously this was a one-off choice when the streamline or line (or circle) of streamlines was created.

Macro commands to dump values of object surface contours and object profiles to file have been added.

Solver- Earth

General

The settings of the solver controls LITER(phi) and ENDIT(phi) can now be reset interactively during the course of the run from the monitor interrupt screen.

The settings of the sweep-to-sweep maximum increment can now be reset interactively during the course of the run from the monitor interrupt screen.

For CONWIZ, the initial values of k and ϵ set by Earth in the absence of explicit settings made by the user have been adjusted. They are now:

$$FIINIT(k)=(3.16228E-3*REFVEL)^2; FIINIT(\epsilon)=C\mu C_D*FIINIT(k)^2/\nu_l$$

Where REFVEL = reference velocity, by default 10m/s and

$$C\mu C_D = 0.09; \nu_l = \text{laminar viscosity}$$

With default values, these lead to $FIINIT(k)=0.001$; $FIINIT(\epsilon)=0.009$, giving a turbulent kinematic viscosity equal to the laminar kinematic viscosity.

When a heat source is applied to a PLATE object located on the face of a non-participating BLOCKAGE

(material 198), and some of the affected cells are cut by Parsol, the source applied in the cut cells is many orders of magnitude too high. This is now fixed internally, but could previously be avoided by setting the porosity of the plate to 1.0.

The equilibrium wall functions (GRND2) use a successive-substitution iteration scheme to calculate the skin friction coefficient. Recent experience has shown that for low Reynolds Numbers, the default number of iterations may not be enough to guarantee convergence. Sometimes the scheme would leave an unrealistic value, leading to incorrect Stanton Numbers. The scheme has been replaced by a Newton-Raphson method which does converge within the default number of iterations.

When the LVEL turbulence model is used in conjunction with heat transfer, the treatment of laminar and turbulent skin friction for evaluating the Stanton Number was not consistent with the treatment for other turbulence models. This is fixed.

The operation of the SUN object in parallel has been corrected. An object would only cast a shadow in the domain(s) in which it existed – it can now cast a shadow into neighbouring domains.

InForm

Multiple mass sources in the negative Z direction, with a solid blockage in the Z plane above, would only give the correct behaviour for the second and subsequent sources. The first would not inject the correct negative momentum, even though the RESULT file indicated that it had. This is fixed.

Sparsol

An INLET object could give slightly different mass sources for 'Velocity' than with 'Mass' or 'Volume' even when all three are set to give the same source. This happened when the inlet is not cuboid – for the 'Velocity' method there was a missing multiplication by a factor taking account of the true surface area of the object. This is fixed.

In the case where a thin blockage cut the link between adjacent cell centres in such a way that both cell centres were in fluid zones with the solid in between, the Stanton Number at the far end of the link was calculated using field values at the near end. This resulted in an incorrect thermal resistance being applied. This is now corrected.

Some inconsistencies in the use of the distance from the near-wall fluid cell centre to the surface have been corrected. In some cases the distance based on the grid spacing was being used instead of the actual distance to the wall.

Parsol

An indexing error in the Earth-Generated-Wall-Functions has been corrected. This could lead to run-time failure in some instances, and was the cause which sometimes prevented cases from PHOENICS 2018 running in PHOENICS 2019 v1.0.

VOF

Due to the inadvertent inclusion of an older file, the use of VOF for thermal calculations was not correct. This is now fixed.

Parallel

An error which led to unexpectedly-large velocity differences at processor boundaries has been fixed. The error only applies when the default linear-equation solvers are in use, and was not present in earlier versions. The new linear-equation solvers introduced with PHOENICS 2019 v1.0 were not affected.

The calculation of average in- and out-flow values of scalars at fixed pressure boundaries was only correct the first time the values were printed. Due to a coding error, the internal stores used to hold summed values were not zeroed again, so the printed values continued to rise each time the coding sequence was executed. This error, which was not present in previous versions, is now fixed.

The deduction of the cell location of the maximum correction printed to RESULT was not always correct, depending on which processor contained the maximum correction. This printout is now correct.

For MOFOR, the copying of a local motion file (*.mof) to the slave processors has been restored.

Flair

The fan-matching option has been re-written to be more robust, and to allow for parallel operation. In addition, a user-adjustable linear relaxation factor has been introduced to add extra control to the change in fan velocity as a response to computed changes in pressure difference across the fan.

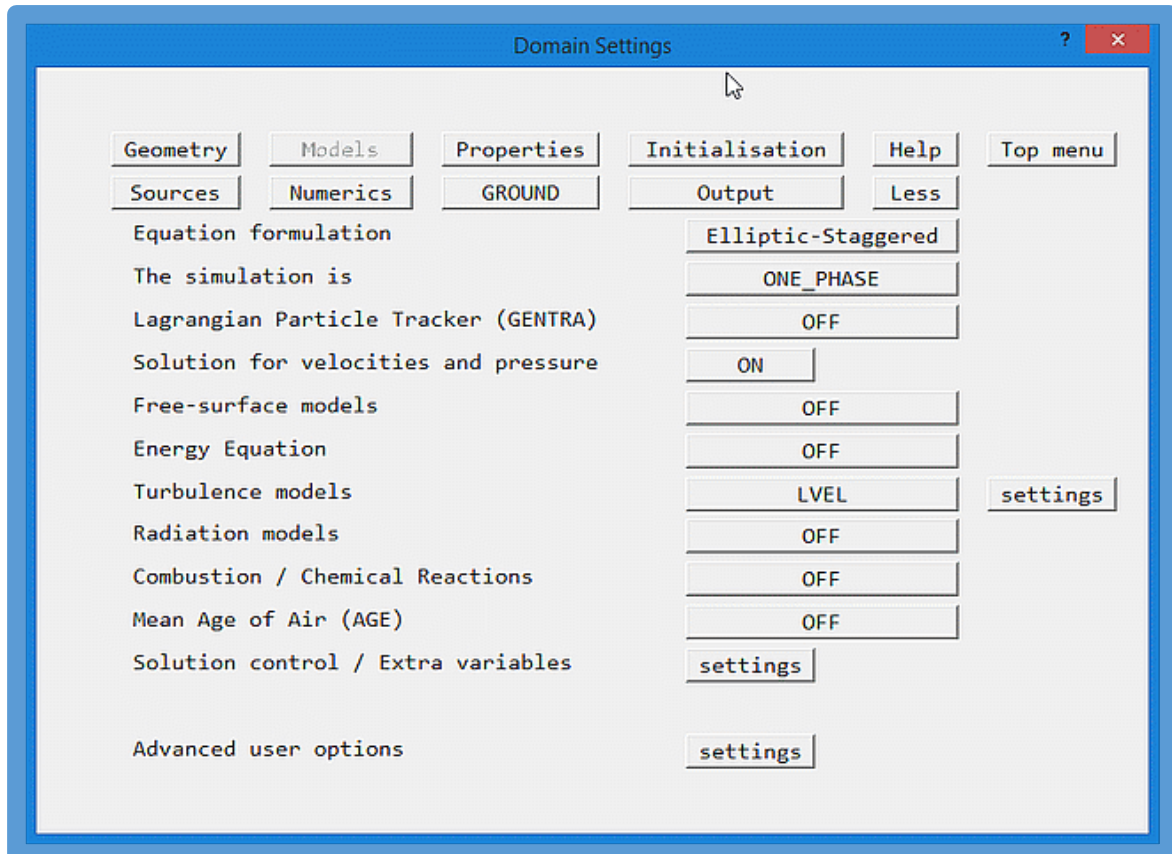
In parallel, any local FANDATA file (containing the fan characteristics) is now automatically copied to the slave processors.

Items in PHOENICS 2019 v1.0 (March 2019)

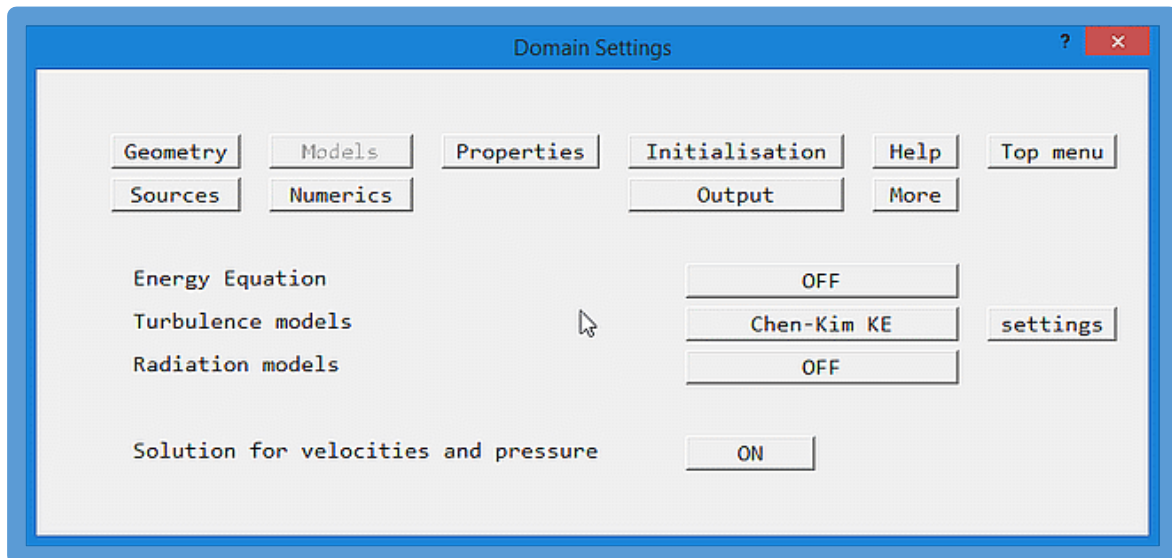
VR-Editor

Simplified Menus

The VR-Editor Main Menu panels can now operate in two modes, (show) Less and (show) More. (Show) Less is equivalent to the previous versions, in that all available options are presented. This may be daunting to new users who only wish to see the most commonly used options. This is the (show) More mode. The first image shows the Main Menu Models panel in the default (show) Less mode:



The same panel in (show) More mode is:



The mode is recorded in the Q1 file saved at the end of the session, and hence will be used next time the

Q1 is loaded.

New Radiation Model

The dialogs for selecting radiation models have been extended to include the new P1-T3 model, described [below](#) in the Solver section.

New Boundary Condition

The dialogs for setting boundary conditions on faceted non-participating blockages (material 198) have been updated to allow the setting of thermal and scalar sources on the outer surfaces of such objects. More details are provided [below](#) in the Solver section.

Extensions to the Volume-Of-Fluid (VOF) method

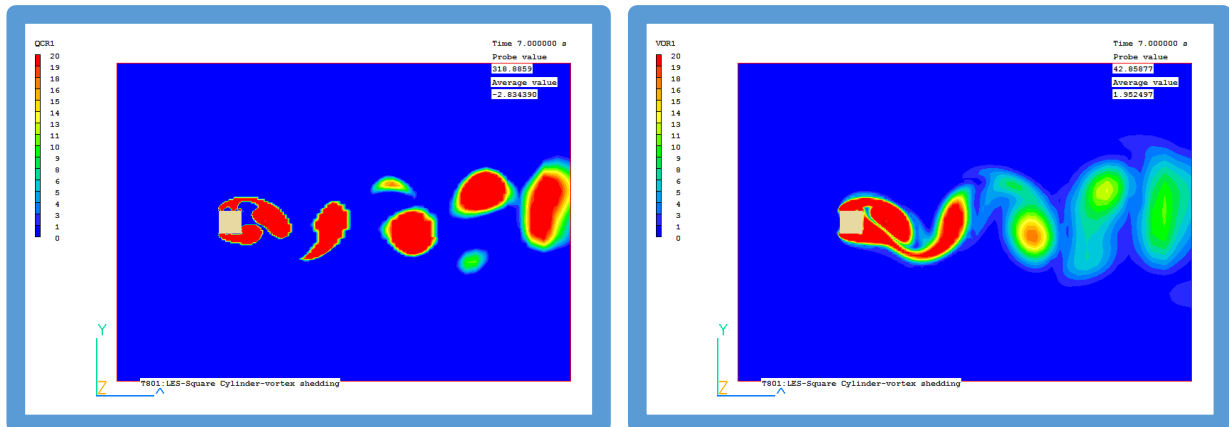
The dialogs dealing with VOF have been updated to enable the selection of the extra THINC method, and to set the parameters required for temperature-dependent surface tension. More details are provided [below](#) in the Solver section.

Additional Linear Equation Solvers

The Main Menu – Models – Solution control / Extra Variables panel has been updated to allow the selection of additional solvers and pre-conditioners for each variable. More details are provided [below](#) in the Solver section.

Activation of Q-Criterion and Vorticity

The Main Menu – Output – Derived Variables panel has been updated to allow the activation of the calculation and storage of the Q-criterion and the Vorticity. The following images show Q-criterion and Vorticity contours during vortex shedding from a square block:



User Interfaces for Service Programs

Graphical User Interfaces have been provided for the PHISUM and PHIDIFF programs. PHISUM is used to sum values from a sequence of solution files, usually to derive an averaged wind speed over a number of sectors. PHIDIFF is used to subtract two solution files to create a third. This can be very useful in locating differences between two solutions. Both programs are activated from Run – Utilities.

Multi-Run

The multi-run feature has been enhanced to allow multiple simultaneous runs.

HandSet Positioning

If the handset is placed in the default location – at the upper-right of the main display window – it will retain that relative position when the size of the main window is changed. If it is moved away to some other location, it will stay there when the main window size is changed. Moving it back to the default position will make it 'stick' again.

Flair / Flair EFS

Wind Attenuation Coefficient

The Wind Attenuation Coefficient is defined as the modelled speed divided by the speed in absence of geometry minus 1.

$$WAT = (\text{absolute velocity} / \text{wind speed at local height}) - 1$$

The derivation of this coefficient is activated from the WIND object dialog, and the values are placed in the 3D store WAT for display in the Viewer.

AIRVOL as synonym for ROOM

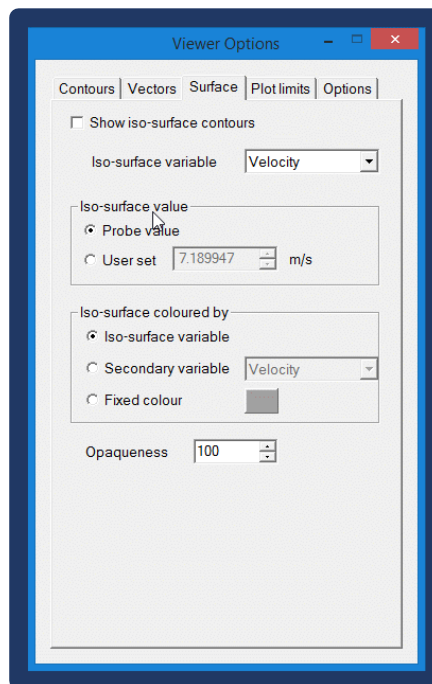
The existing ROOM object is used to define a volume of space. The Earth solver will calculate the volume of the object, excluding any solid obstructions, and sum up the volumetric inflow rate across the boundary of the object, including any internal air supply. It will then print into the RESULT file the number of **Air Changes per Hour** for the volume defined by the object. This is calculated from $3600.0 * (\text{sum of volumetric inflows}) / (\text{room volume})$. The ROOM object has no effect on the solution.

As this function can apply to any open or closed volume, not just a room, the synonym AIRVOL has been introduced to make this clearer to users.

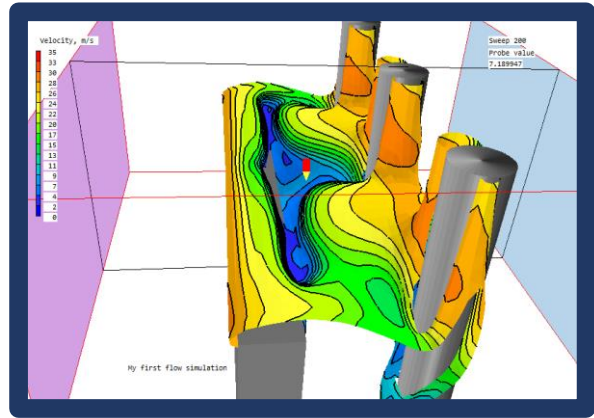
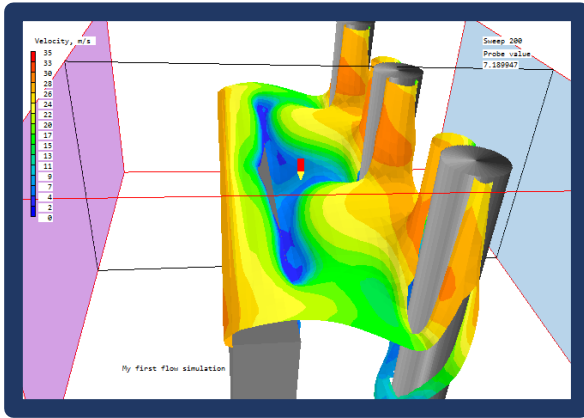
VR-Viewer

iso-surface Update

The iso-surface contour feature has been extended to allow an iso-surface of one variable to be coloured by the contour map of another. This is activated from the 'Surface' tab of the 'Viewer Options' dialog,



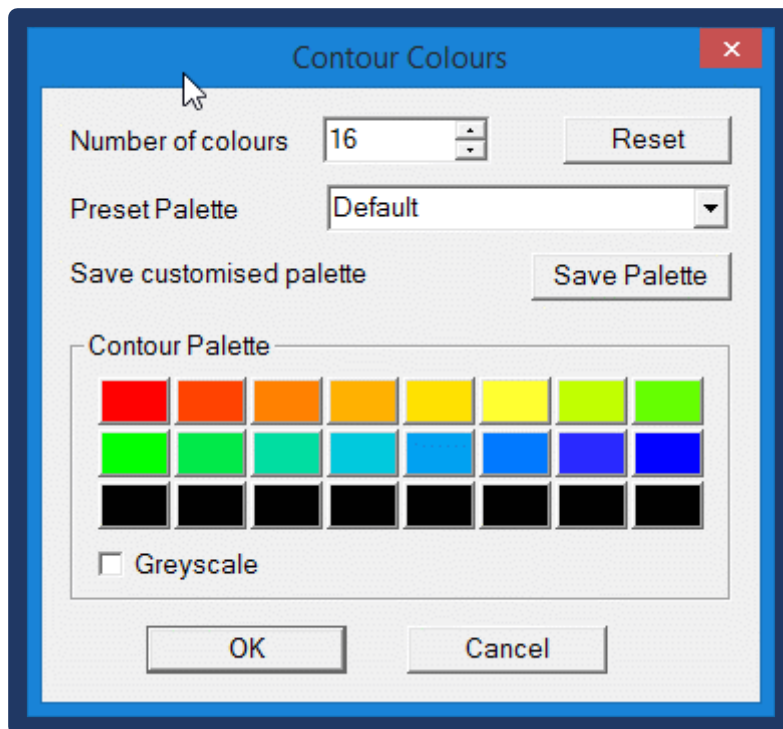
The 'secondary' variable can be any SOLVED or STORED variable. The image below shows an iso-surface of P1 coloured by Velocity:



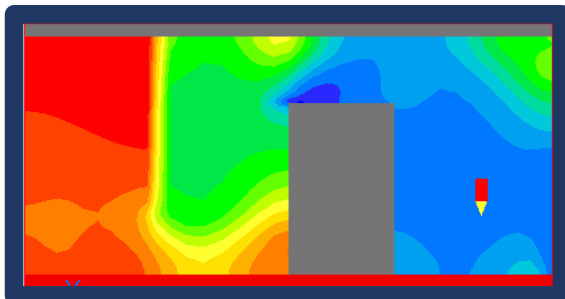
The 'Fill', 'Lines' and 'Fill and lines' options on the 'Contour' tab also operate on the iso-surface contour map.

Choice of colour maps

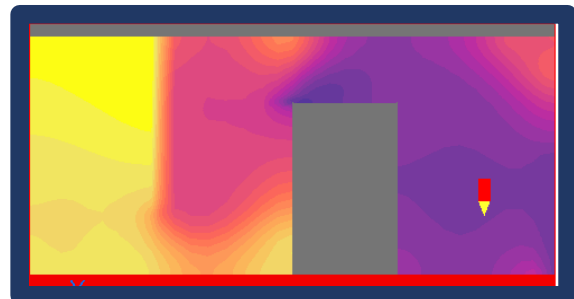
Several pre-set colour maps have been made available. These are selected from the 'Update palette' button on the 'Contours' tab of the Viewer Options dialog.



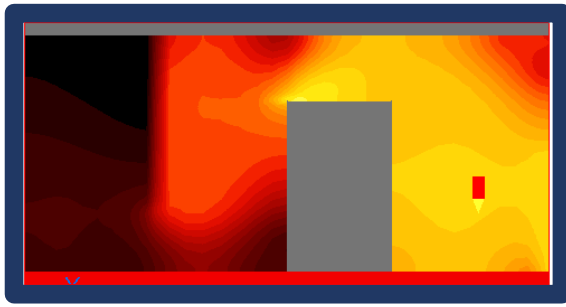
The dialog allows one of several pre-set colour palettes to be loaded. These are:



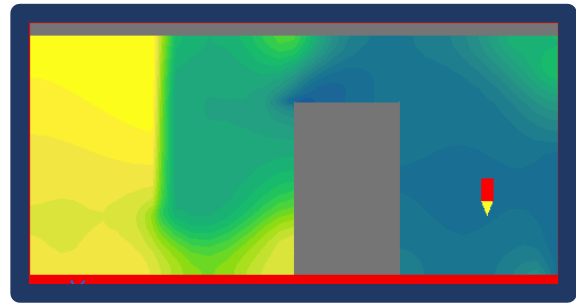
Default CHAM red-blue



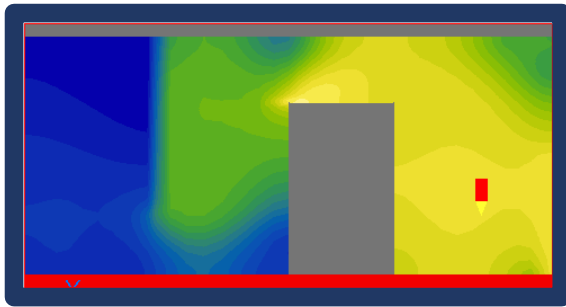
CHAM heat



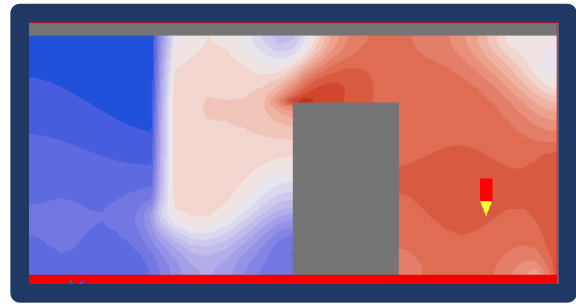
L3 heat



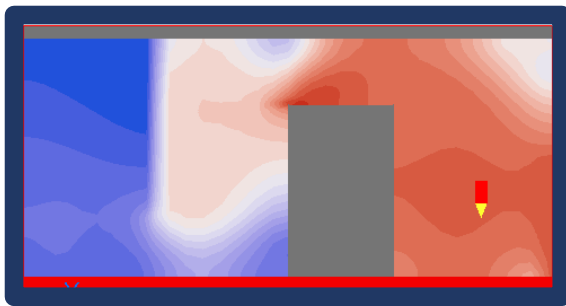
CHAM blue to yellow



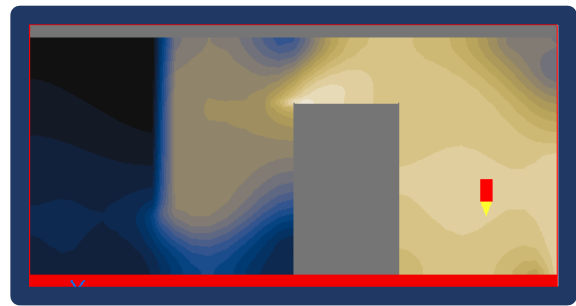
L9 blue to yellow



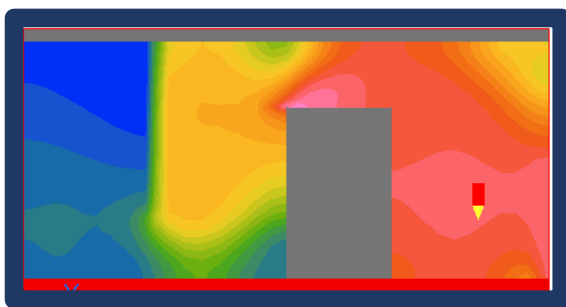
CHAM diverging



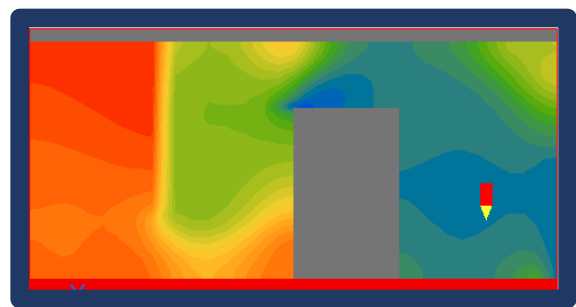
D1 diverging



CBL1 colourblind



R1 rainbow with pink



R2 rainbow without pink

There is also a 'Custom' option, whereby the user can set each colour in the palette, which can then be saved.

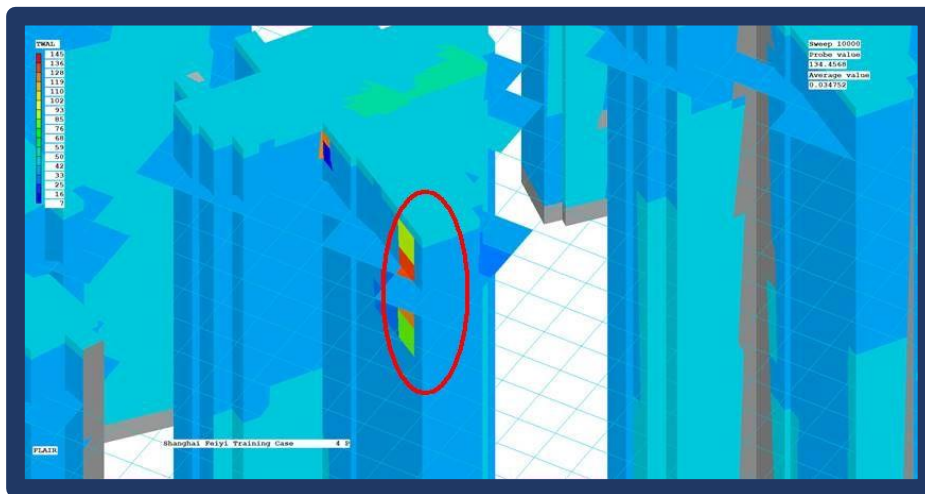
Improved surface contours

In order to plot contours on the surface of an object, the surface the contour is plotted on must be offset slightly from the actual surface of the body. The algorithm for offsetting the plotting surface has been improved, leading to nicer surface contours.

The first image shows surface contours not matching the in-plane contours:




The next image shows the same plot after correction. The mis-alignment is much reduced.



Auto-update of Screen Image

A click on F9 causes the Viewer to refresh the screen image from the latest solution file written by the Earth solver. A right-click automates this process, and the screen image will be refreshed each time the Earth solver writes a solution file.

Stopping an Animation

Users have reported that it can be difficult, if not impossible, to stop an animation early especially if the solution files are large and it takes a long time to read each file. The mechanism for checking for an interrupt (pressing the animation toggle  again) has been rewritten to make this process more reliable. In addition, if VR-Viewer finds a file called 'stopit' or 'stopit.txt' in the working folder during an animation, it will stop the animation and delete the file.

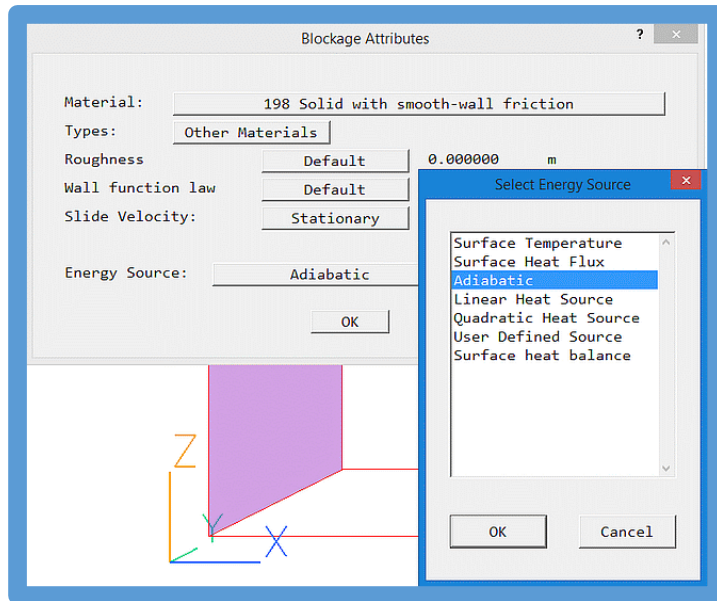
Solver

New P1-T3 Radiation Model

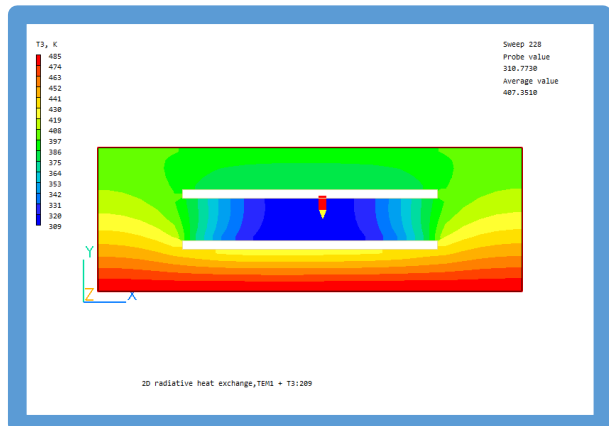
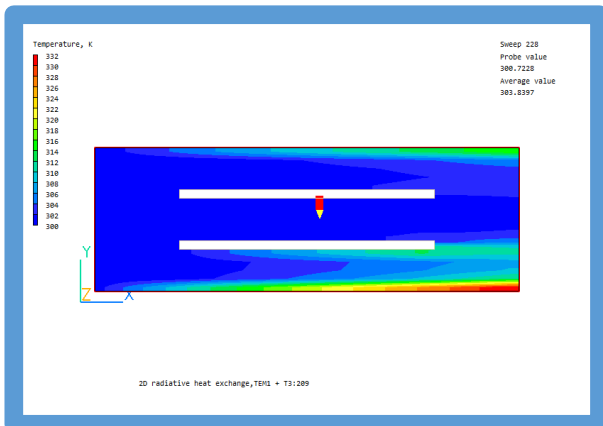
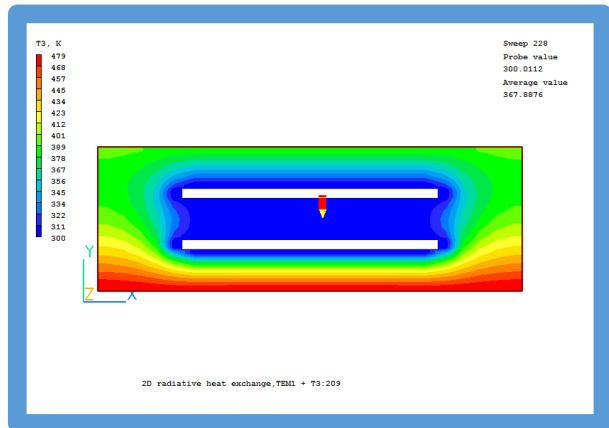
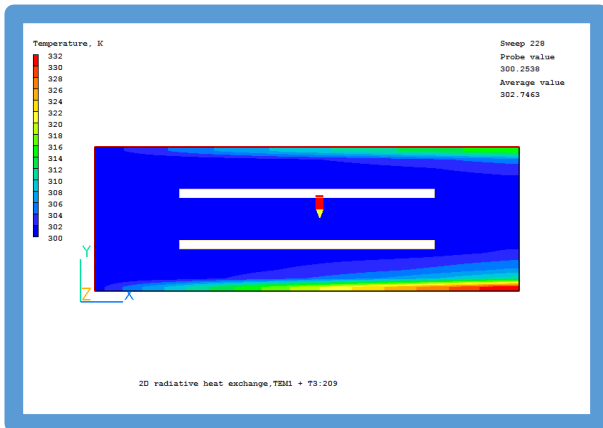
This is the P1 Radiation Model, cast in terms of the radiant temperature, T3 rather than the irradiance. Because of this, the model has all the links to conjugate heat transfer that the IMMERSOL model does. It is the IMMERSOL model without the interwall-distance terms, using the Marshak condition at gas-solid interfaces. For this reason it is superior to the existing Radiosity model which has no conjugate heat transfer capability. The model is only suitable for optically-thick situations, for example when combustion is active.

New Boundary Conditions for Non-Participating Blockages

It is now possible to set a range of heat and scalar boundary conditions on the outer surface of a faceted non-participating blockage (i.e. one using material 198 and does **not** use a geometry starting 'cube').



When the IMMERSOL or P1-T3 radiation models are active, the 'Adiabatic' condition is now implemented in the same way as at adiabatic plates – the sum of the convective and radiative fluxes is zero, rather than each flux being zero as was previously the case. A surface temperature (stored in TWALL) is deduced, and is used as the thermal and radiative boundary condition. As an illustration, the following images show Temperature and T3 contours for flow past two adiabatic blocks, first from PHOENICS 2018 and then from PHOENICS 2019.

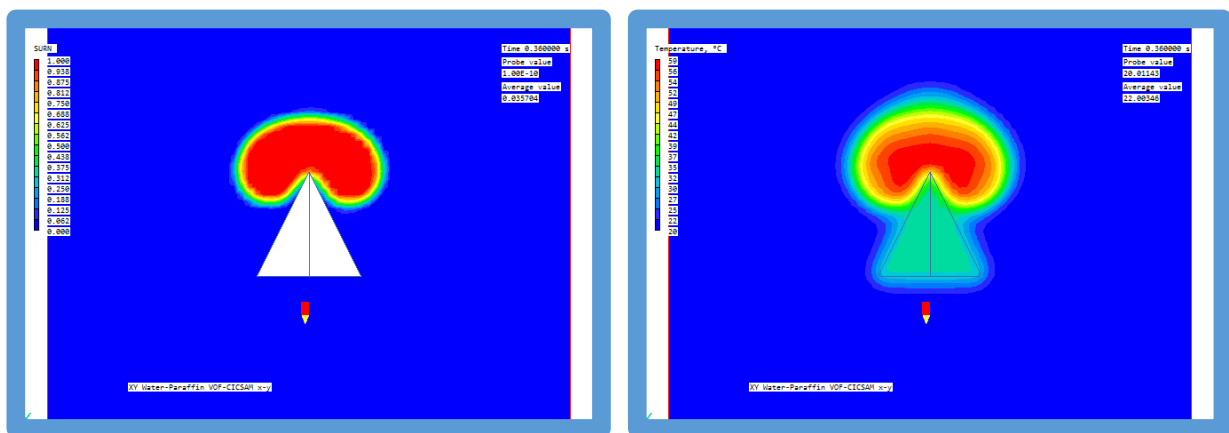


The ‘Surface heat balance’ option is also new. The surface temperature is deduced from a heat balance between the fluid temperature in the cell adjacent to the surface, and a user-set fictitious internal temperature and internal resistance. The heat flux is then computed based on the surface temperature and an external heat transfer coefficient obtained from the wall function, or a user-set constant.

Extensions to Volume-Of-Fluid (VOF)

The **THINC-WLIC** method has been implemented in PHOENICS, as an additional VOF method. THINC (tangent of hyperbola for interface capturing) uses the hyperbolic tangent function to devise a conservative, oscillation-less and smearing-less scheme. This scheme shows competitive accuracy compared to most existing methods without any geometry reconstruction. Multi-dimensional computing is conducted by the WLIC (weighted line interface calculation) method.

All the VOF methods can now solve temperature-dependent cases, with proper treatment of the temperature in each phase and in any immersed solids. The following images show a drop of hot fluid falling over a solid obstacle:



Options have been added to make the surface tension a linear function of temperature, or to use the Langmuir equation of state which includes a scalar as well as the temperature.

A constant static contact angle can be specified to model wall adhesion effects.

New Linear Equation Solvers

Many additional linear-equation solvers and pre-conditioners have been incorporated, including several from the [HYPRE](#) suite.

The following linear equation solvers are available:

1. STONE - default PHOENICS Stone-type solver, which does not use a pre-conditioner
2. CGRS - Conjugate-Gradient-Residual Solver
3. CNGR – existing PHOENICS Conjugate-Gradient-Residual solver - with ILU0 or JACOBI pre-conditioners
4. MSIP – existing PHOENICS MSIP
5. BCG - Bi-Conjugate Gradient Method
6. BCGSTAB - Bi-Conjugate Gradient Method stabilized
7. TFQMR - Transpose-Free Quasi-Minimum Residual method
8. FGMRES - Flexible version of Generalized Minimum RESidual method
9. DQGMRES - Direct versions of Quasi Generalize Minimum Residual method
10. AMG - BoomerAMG from HYPRE, which does not use a preconditioner
11. HBiCGStab - BiCGStab from HYPRE
12. HGMRES - GMRES from HYPRE
13. HFlexGMRES - FlexGMRES from HYPRE

14. HLGMRRES - LGMRES from HYPRE

Note that in parallel, the STONE, CNGR and MSIP solvers are replaced by a parallel version of the CGRS solver. The remaining solvers operate in serial and parallel modes.

The following preconditioners are available:

1. NONE - preconditioner is absent
2. PBP - point-by-point preconditioner
3. Jacobi - iterative Jacobi preconditioner (CNGR only)
4. ILU0 - simple ILU(0) preconditioning (CNGR only)
5. AMG - BoomerAMG as preconditioner from HYPRE
6. ParaSail - ParaSail from HYPRE
7. Euclid - Euclid from HYPRE
8. PILUT - PILUT from HYPRE

Note that not all preconditioners are available for all solvers. The Stone, MSIP and AMG solvers do not require a preconditioner.

The HYPRE solvers can only use PBP or HYPRE preconditioners.

The solvers and pre-conditioners can all be activated on a pre-variable basis from the Main Menu – Models – Solution control / Extra variables panel.

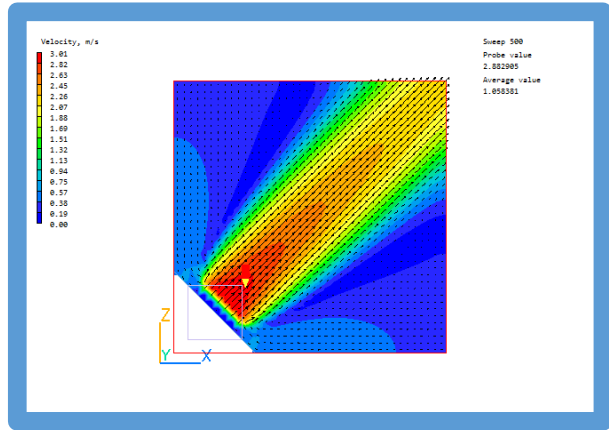
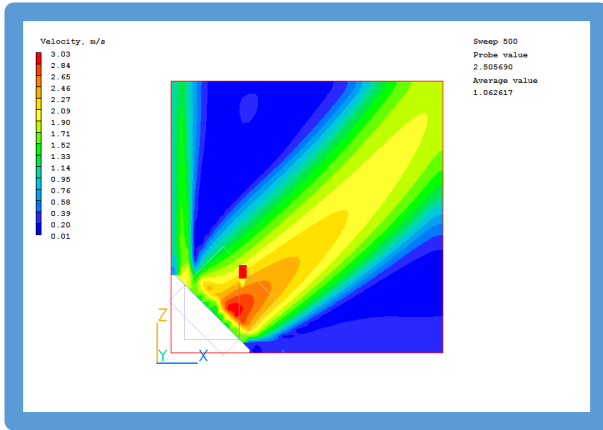
The existing PHOENICS solvers, both in sequential and parallel mode, have to solve over all cells in the domain, regardless of whether the cell is blocked or not. When a large proportion of the cells is blocked, convergence of the solver may be adversely affected. In parallel, load balancing may be hard to achieve, as some processors may have few or no active cells.

The new solvers and pre-conditioners differ in that they only load 'live' cells into the solver matrix. This can greatly reduce the workload of the solver, at the expense of having to copy coefficients into the solver matrices, and then to copy the solution out. There is also a load balancing algorithm for parallel operation which ensures that all processors have to solve approximately the same number of equations regardless of the blockage distribution.

The new solvers, especially the ones from HYPRE are less affected by grid refinement, and should prove superior for fine grids, especially when there are many blocked cells.

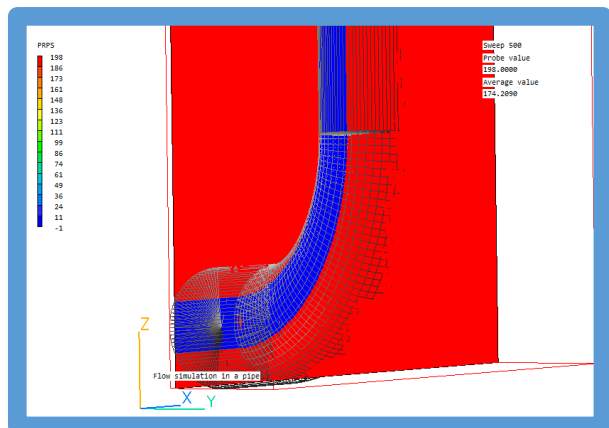
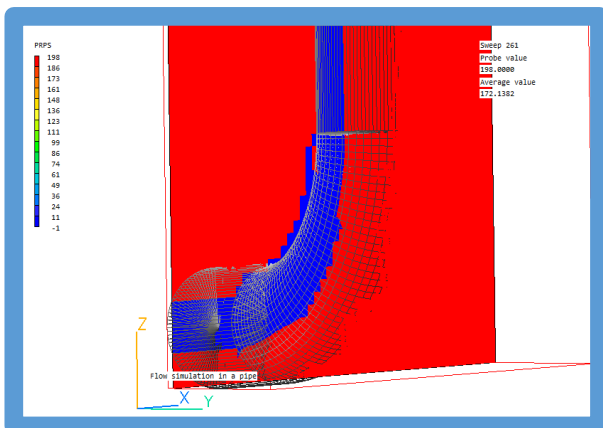
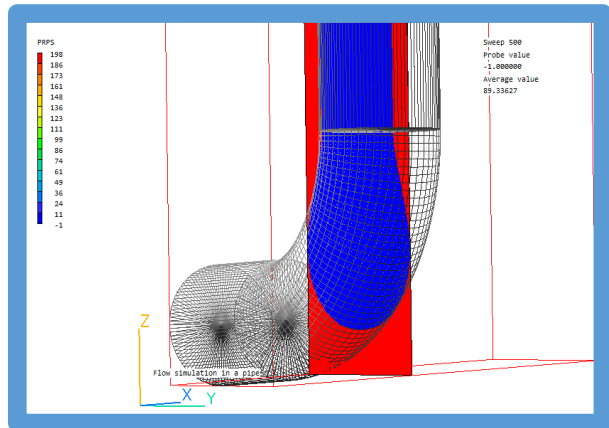
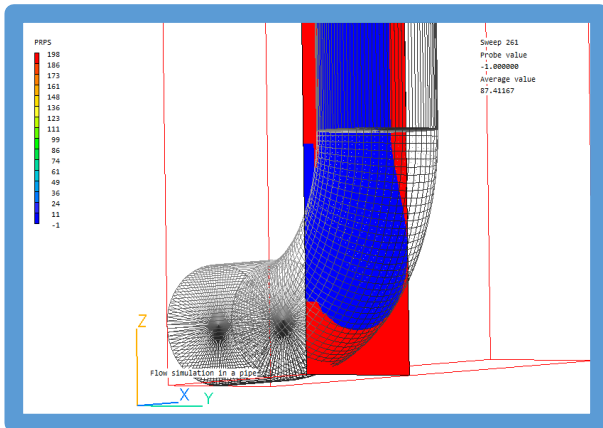
SPARSOL – Angled-In

The treatment of ANGLED-IN objects by SPARSOL has been improved, as shown the following two images. They show an ANGLED-IN acting on the surface of a wedge blockage, firstly in PHOENICS 2018, and secondly in PHOENICS 2019.



PARSOL Upgrade

In PHOENICS 2019 SPARSOL remains as the default cut-cell method. The alternative PARSOL method has been substantially rewritten, with the ultimate goal of allowing for double-cut cells. At the moment this feature is disabled, but the quality of detection has been improved. The next images show the PRPS fields produced for Library case 627 by PHOENICS 2018 on the left and PHOENICS 2019 on the right.



The new PRPS fields are much smoother and fit the geometry better.

Choosing 'Old-PARSOL' as the cut-cell method on the 'Grid mesh settings' dialog will cause Earth to use the 2018 Parsol for users who wish to reproduce previous results as closely as possible. This can also be achieved by adding ISG60=1 in group 19 of Q1.

Thin Plate

The THINPLT coding has been revised again, to get the plate temperature directly from a heat balance including the transient term, and then deduce the surface temperatures. The surface temperatures are

now held internally in patch-wise storage so the TWAL variable becomes optional. If present, TWAL will be filled with the local surface temperature at THINPLT, PLATE and BLOCKAGE objects, whenever it is deduced from a heat balance.

Reduced Convergence Report

At the end of a run, a brief report showing the level of convergence achieved is displayed using the default system browser.

Minor Corrections / Bug Fixes

The transient contribution to mass balance for GALA is calculated and printed.

Problem with InForm (source of var is coval(..,..)) when there are several InForm coval sources for the same variable at the same location but with different 'with if()' qualifiers fixed.

Catch 'disk full' error when writing RESULT, and allow user to make space.

Fix error in InForm, where (property of ...) setting only worked on first step of transient.

Corrections to writing transfer object output files, ensure 0.0 is written in cells outside the facets of the object.

If an existing InForm table file is not found on restart, write the table headers.

Skip checking cells with material 198/199 when gathering minimum and maximum values in the domain.

In wall-function code, don't average face velocity to cell centre if forward or back face is blocked. This is consistent with velocity-averaging elsewhere.

In parallel, issue error message if target of InForm SUM operation cannot be found on any processor.

In parallel, issue error message if there are not enough cells for the chosen manual decomposition.

CVD

The maximum number of allowed surface reactions has been increased to 150.

The error messages issued when there are errors in the reaction database files have been improved to be more informative.

PIL

New logical variable P1T3 to signal that the P1-T3 radiation model is active when T3 is solved rather than IMMERSOL.

New integer variable ISRFTA to select the surface tension formula.

New real variables SURFTB and SURFTC to pass constants to the surface tension formula.

```
If      ISRFTA=1;     $\sigma = SURFTA$ 
        ISRFTA=2;     $\sigma = SURFTA + SURFTB * (TEM1 - SURFTC)$ 
        ISRFTA=3      $\sigma = SURFTA * (1. + SURFTB * (TEM1 + TEMPO) * LOG((1.0 - Cn) / SURFTC)$ 
```

Where Cn is one of the concentration variables.

New logical VOFcnt to signify contact angle active, and CNTANG to set the angle.

New real variables CFLl, CFLU, BTATHC, UUDF required for VOF – THINC.

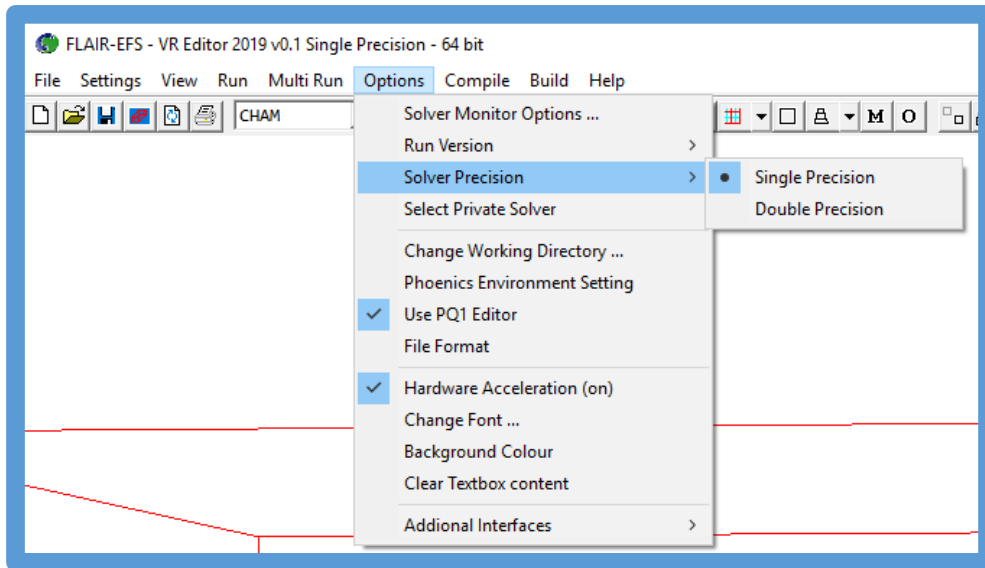
Generic

Directory Structure

In addition to the existing d_win32 and d_win64 directories which contain the single-precision 32-bit and 64-bit code, there are now d_windble directories which contain the 64-bit double-precision code. No provision has been made for 32-bit double-precision.

Activation of Double-Precision

The activation and implementation of double-precision has been changed. The selection between single- and double-precision is now on the VR-Editor Options menu:



Once Double precision has been selected, the 'Compile' and 'Build' commands will compile and build double-precision versions of the solver.

Unlocking

Error messages relating to problems with unlocking are captured in the text file `unlock_error.txt` so that users do not have to rely on screen-grabbing the error message.

Installation

The installer for PHOENICS 2019 performs a pre-requisites check to ensure that any items needed to run PHOENICS are present. This will eliminate the problem some users have encountered where some Microsoft DLL files are missing.

Parallel

The MPICH2 software used in previous versions of PHOENICS is now quite old, and due to changes made by Microsoft to the Windows Registry, it can be impossible to install it on modern Windows 10 machines. As a result, PHOENICS 2019 will be shipped with MS-MPI.

Items in PHOENICS 2018 v1.3 (November 2018)

VR-Editor

Correction to potential error when leaving BFC mesh generation menu. Entering the 'Modify time step distribution' would cause the display of the mesh to be disrupted on leaving the dialog..

Flair / Flair-EFS

Add WAT Wind Attenuation Coefficient, defined as:

$$\text{WAT} = (\text{absolute velocity} / \text{wind speed at local height}) - 1$$

Add AIRVOL as synonym for ROOM. The AIRVOL (AIR VOLUME) object behaves exactly as the ROOM object, and produces the same outputs.

Correction to writing domain size toggle to Flair-EFS Q1. If the domain size was changed from 'Factor' to 'Total', this was not captured to Q1.

Corrections to Drift Flux Model to prevent crash when the deposition integral is zero.

VR-Viewer

When the Viewer is drawing animations from large files it can be hard to interrupt and stop the animation early. If Viewer finds a file called 'stopit' or 'stopit.txt' it will stop the animation.

Surface contours are improved, especially for large domains, by basing the offset between the object surface and the surface contour on the local cell size rather than the distance from the domain origin.

Solver

Correction to THINPLT for use of emissivity. The 'Low' side emissivity was always used for both sides of a THINPLT object.

Correction to Algebraic Slip Model to prevent overwrite when number of inflow/outflow patches is greater than the number of particle phases.

CVD

Maximum number of reactions and gas species each increased to 100.

Generic

PHOEPATH.BAT updated to allow for Version 19 of Intel compiler.

Compile scripts modified to enable build script to work even when PHOENICS is not installed at the root of a drive.

Items in PHOENICS 2018 v1.2 (July 2018)

VR Editor

Labels added for the emissivity of both sides of a THINPLT object on the object Attributes dialog.

Correction to the reading of inlet profile file in the WIND object to eliminate the need for dummy blank lines at the end of file.

VR-Viewer

Move status display message when reading cut-link data outside the loop reading each link to speed up the reading of the solution file.

Corrections to the saving animated GIF files.

Solver

Correction to the reading of inlet profile file in the WIND object to eliminate the need for dummy blank lines at the end of file.

Stop parallel run when HOL is active, as not compatible.

Flair-EFS

Correction when opening case from Flair-EFS 2015.

Generic

Corrections to treatment of old USB dongles.

Support for Sentinel dongles removed.

Items in PHOENICS 2018 v1.1 (March 2018)

VR-Editor

Thin Plate Object

When a THINPLT object detects that the energy equation is being solved, it automatically STOREs the 3D variable TWAL. This will be filled by the Earth solver with the surface temperatures of each side of the thin plate. The Editor now allows the emissivity of each side to be set separately.

Solver – Earth

Thin Plate Object

The treatment of the THINPLT object has been updated. Instead of the optional 3D variable TWAL containing the average temperature of the thin plate, TWAL is now required and contains the surface temperature of each side. This allows for better energy conservation when the plate lies in the X-Y plane whilst allowing for a smooth restart. TWAL values inside solid blockages will print to RESULT and be visible in the Viewer.

The thin plate treatment when solving for enthalpy (H1) has been made consistent with that used when solving for temperature (TEM1).

Transient Term in Energy Equation

The re-arrangement of the time-flux summation for the energy equation introduced for Version 2018 v1.0 is strictly only correct for incompressible flow. The term dealing with mass change due to density difference was erroneously omitted. This has been fixed.

In parallel, the gathering and summing of energy time-fluxes was not correct. It is now fixed.

Note that in both cases it was only the time fluxes printed to RESULT which were incorrect, the source used in the solution was correct.

Private Solver under Windows 10

It has been reported by users that when a private earexe.exe is built using Studio 2012 or later under Windows 10, then the build will succeed and generate an earexe.exe but this executable will not start properly under Windows 10. If the same executable is copied to a machine running Windows 7, it will run normally.

The problem has been traced to the libraries used for checking the license file, and has been fixed by using more recent versions of the libraries.

Parallel Bug Fix

For 2018v1.0 a small economy was introduced by not solving for the wall-distance variable (LTLS) when there were no walls (PLATEs) or obstacles (BLOCKAGEs). In parallel, it is possible for one processor to satisfy this condition, but another not. In that case the solution will hang as the processors lose synchronisation. This has been fixed by summing the number of walls over all processors before deciding to skip the LTLS solution.

InForm MODCON Command

In all previous versions, the InForm command **MODCON**(var at name **with HIGH**) did not zero the low convection flux of the plane above, but the low flux of the current plane. This has been fixed.

Items in PHOENICS 2018 v1.0 Added since the last release of PHOENICS 2017 (dated March 2017)

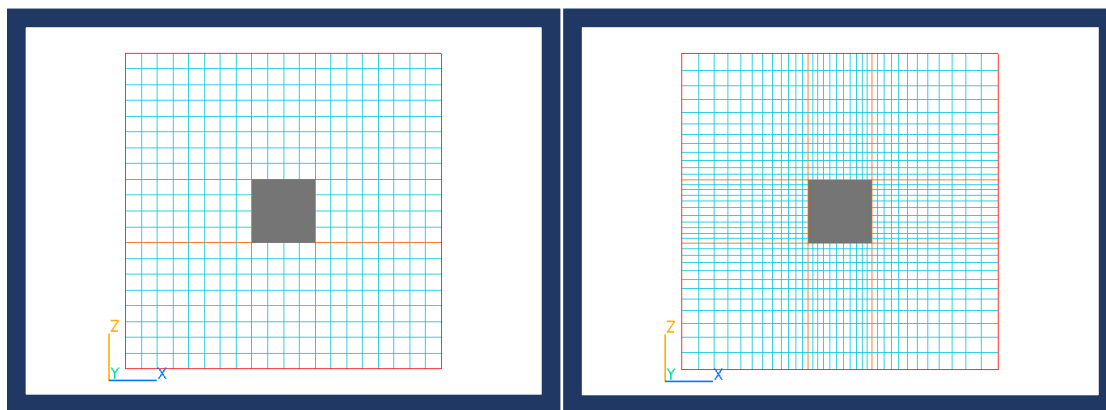
VR-Editor

Change to Automesh

The default Automesh settings have been changed to make the behaviour of the automesher more predictable. Instead of starting with a cell size $1/20^{\text{th}}$ of the domain, the initial mesh is always 1 cell per region. This is then refined until either the:

- ratio of cell sizes across all region boundaries satisfies the set criterion, or
- 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.

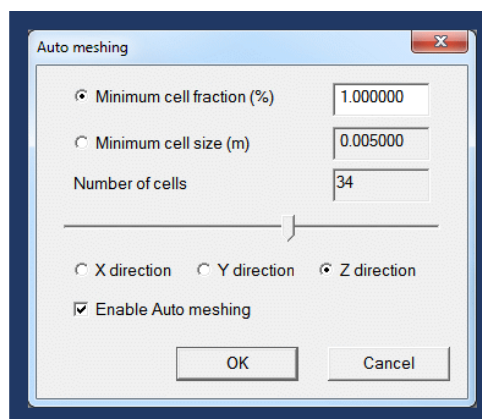


The image on the left shows the old automesh, that on the right the new, both with default settings.

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 the minimum cell size, so increasing the number of cells; conversely moving the slider right increases the minimum cell size so decreasing the total number of cells.

For compatibility with existing cases, the new rules are only applied to new cases, unless the '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

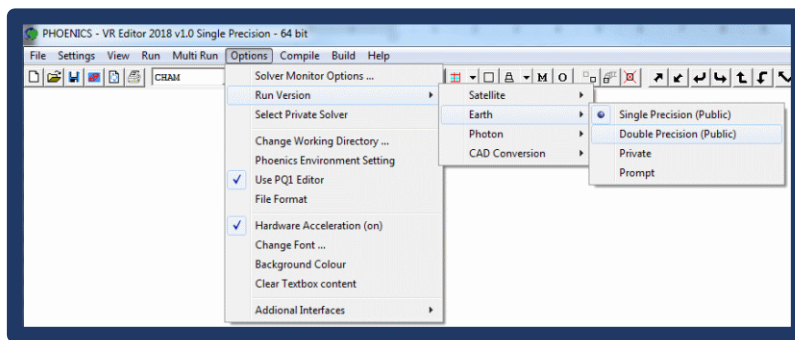
It is now possible to set an additional source for scalars passing through a pair of linked ANGLED-IN objects. One can:

- 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.

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.

Activation of Double-Precision Solver

A double-precision Earth solver has been delivered for some time, but the activation relied on knowing where it was, so that it could be selected as the 'Private' solver. (For information, it was and is in \phoenics\d_earth\d_win64\earthble.exe). It is now selectable directly from the 'Options / Run Version' Menu.



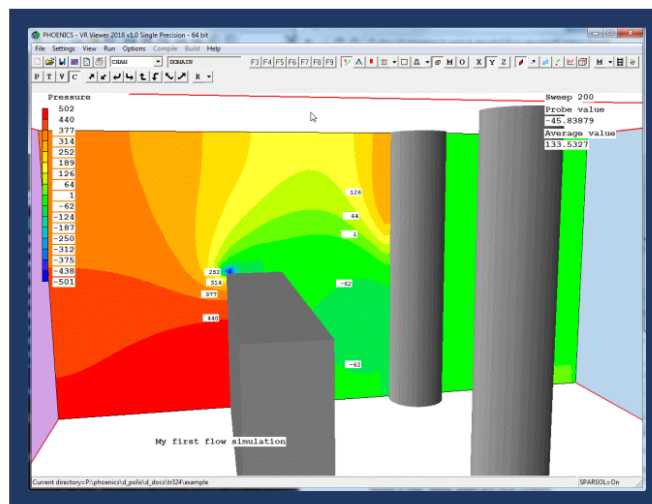
Designate rotating objects for ROTOR

The algorithm for detecting which objects should rotate with a ROTOR object is not always 100% reliable. The user can now manually choose which blockage objects are rotating.

VR-Viewer

Labelling of Contours

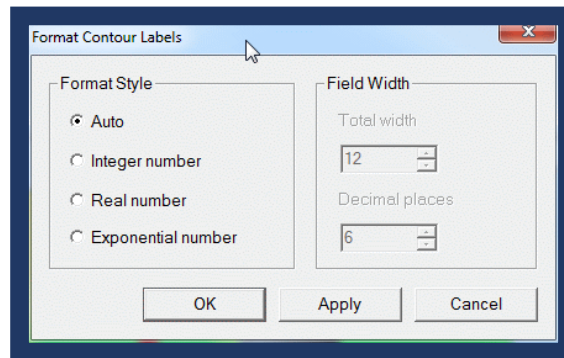
The lines separating the contour bands can be labelled, either automatically or manually.



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

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

- Real
- Exponential



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

Minor corrections to Editor and Viewer

Don't set COVALs for variables which are not STOREd.

Saved STL files (Options – Additional interfaces) don't have leading spaces on each line.

Correction to unit symbols displayed on OPENING and ANGLED_OUT dialogs for deduced external velocity.

Correction to formula used for initial velocity profile in WIND object. Does not affect the final solution, as only an initial guess.

On exit from the 'Domain Edges' dialog, if a WIND condition is selected but a WIND object does not exist, create one with default values.

Set the correct default geometries for domain edge inlet/outlet/plate objects if their attributes have been set.

Don't solve the Laplace grid equations (MAGIC) when reading a BFC case into the Viewer, as the grid will be read from the grid file in any case.

Correction to plotting PRPS for Sparsol to try and avoid occasional white spaces in contour.

Correction to memory handling on exit from Viewer to prevent occasional hang.

When running a Viewer macro, don't switch off all clipping planes, but leave as they were.

Streamlines started 'Along a line' or 'Around a circle' are now re-generated for each step of an animation.

If Sparsol and MOFOR detected when reading PHI(DA) in Viewer, switch to Parsol as this is what the Earth solver will have used.

Q1 – PIL

New PIL variable **ISCHM** – denotes the first sweep 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 takes the following meanings:

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

Any existing SEM case can be converted to VOF just by adding ISURFA=n to the Q1, where n chooses the VOF model. These are described below.

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

Solver – Earth

New and Upgraded Turbulence Models

The following additional 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 foregoing models are made available in both high- and low-Reynolds-number form, and they can be activated through the VR Editor interface, or by the TURMOD command in Q1. The constants for all turbulence models can be reset from Q1 using SPEDAT commands, and for each simulation the values used and model name are echoed at the top of RESULT.

Extensive testing and validation of these models has been carried out on a large number of library cases, including: parabolic and elliptic free jets; Couette, channel & pipe flows, impinging jets, atmospheric boundary layers, buoyant convection in a tall cavity, flows with separation such as the backward facing step, the abrupt pipe expansion with heat transfer, flow through an orifice plate, flow past a blunt plate, flow over a surface-mounted square rib, and flow over a surface-mounted cube.

The option to include buoyancy terms in turbulence transport equations is provided for all k- ω models.

Some minor corrections have been made to two components of the strain-amplification tensor in the Realisable k- ϵ model, which was introduced in PHOENICS 2017.

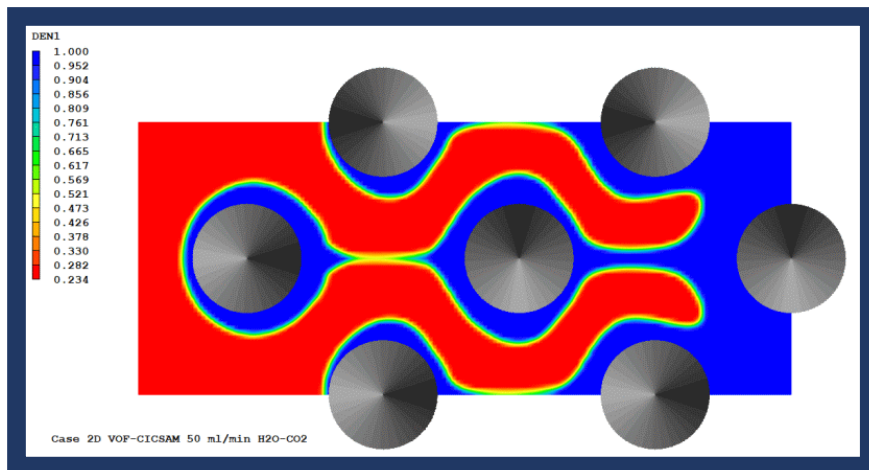
New Free Surface Models

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 is dependent on the value of the Courant number. For Courant number smaller than 0.3 CICSAM should be preferred. For values of Courant numbers below 0.5 and above 0.3,

the HRIC and modified HRIC could be used. For higher values of Courant number, the STACS and MHRIC should be preferred. For large domain (like flow around ships), HRIC and MHRIC should be preferred.



All the above models, as well as the existing HOL and SEM methods can include the surface tension force, using the Continuum Surface Force Method.

Update to Sparsol

Sparsol has been prone to occasionally produce spurious 'beams' of solid, often emanating from a badly-formed geometry element such as missing or back-to-front facets. These have now been eliminated.

The default SPARSOL tolerance has been changed from 5E-4 to 1E-5.

Summation of Transient Term for Energy Equation

In transient cases, the RESULT file contains the 'time-fluxes' printed as 'Transient contribution'. For the energy equation, these time fluxes can be very big, as they include a contribution from $\rho V C_p T_0 / dt$ where T_0 is the reference temperature, $TEMP_0$, which is usually 273. The nett contribution is now summed omitting the T_0 terms leading to more accurate nett values.

For IMMERSOL, the transient term in solids is split between TEM1 and T3, in line with the existing practice for conduction. Previously, the whole term was added to TEM1 and nothing to T3 which could lead to convergence problems.

Upgrade for THINPLT with IMMERSOL and IMMERSOL Heat Balance Reporting

The heat fluxes on either side of a THINPLT object are now printed separately, as is the transient term. In addition, the absorption and emission sources for TEM1 and T3 are now printed, giving a clearer picture of the overall energy balance.

The temperature of the THINPLT object can be stored by STORE(TWAL). On restart, the temperatures of the THINPLT objects will be taken from the TWAL values in the restart file.

VTK Output

The VTKVERTEX.VTK and VTKCENTRE.VTK files are no longer produced. They are replaced by PHI.VTK, which is now also produced by parallel runs.

VTK files now recognise polar grids and two-phase cases. Also corrections to averaging next to blockages to be more like Viewer.

TECPLOT Output

The TECPLOT ASCII output file TECDATA.DAT is replaced by the binary TECDATA.PLT on 64-bit systems. On 32-bit systems the ascii file is retained.

CONWIZ

Parallel cases would sometimes show sudden and unexplained divergence of KE when CONWIZ was

active. This has been managed by increasing the maximum increment allowed for KE, but this was only done automatically for Flair. The cause of the divergence has now been traced and removed (the maximum increment limits were not being applied in the 'halo' cells leading to artificial gradients).

As a result of the above correction, the default maximum increment for KE has been increased from $0.001 * VEL_{ref}$ to $0.01 * VEL_{ref}$, where VEL_{ref} is the reference velocity (defaulted to 1.0)

The default maximum increment allowed for TEM1 has been reduced from 1000° to 50° . If a Flair FIRE object is detected, it is still automatically reset to 10° .

General Improvements and Corrections

- Don't solve for LTLs when there are no PLATE or BLOCKAGE objects.
- The local cell Reynolds Number ($V_{abs} * Vol^{0.333} / (v_t + v_l)$) and local Turbulent Reynolds Number (v_t / v_l) can be STORED as CREY and TREY respectively.
- Correction to the Inform SUM function in parallel.
- A local PROPS (property table) file will be used if present.
- A 'tab' character can be used to separate items in wind profile input files.
- In transient cases, the monitor screen shows the sweep number by default.
- In parallel cases, the number of processors in each direction is echoed to RESULT.
- In parallel cases allow objects to be called IMPORT or EXPORT to avoid confusion with object types.
- In transient GENTRA cases, vaporised particles are removed.

FLAIR

Drift Flux Model for Aerosol Deposition

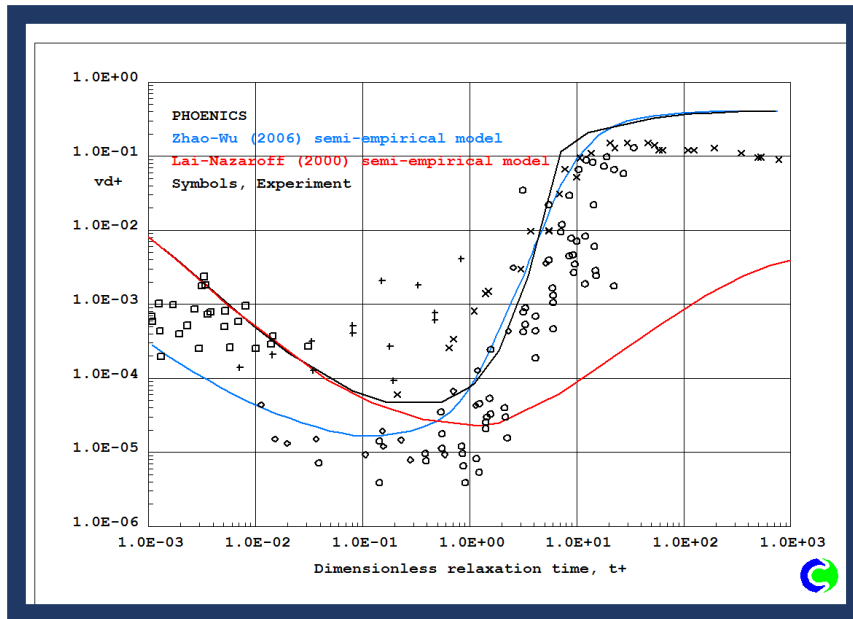
An Eulerian-based multi-phase model for simulating the dispersion and deposition of aerosol particles in indoor environments has been implemented as a standard option in PHOENICS-FLAIR 2018. Typical applications include studying indoor air quality and designing ventilation systems to deal with: human 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 collisions 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 ventilated 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 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

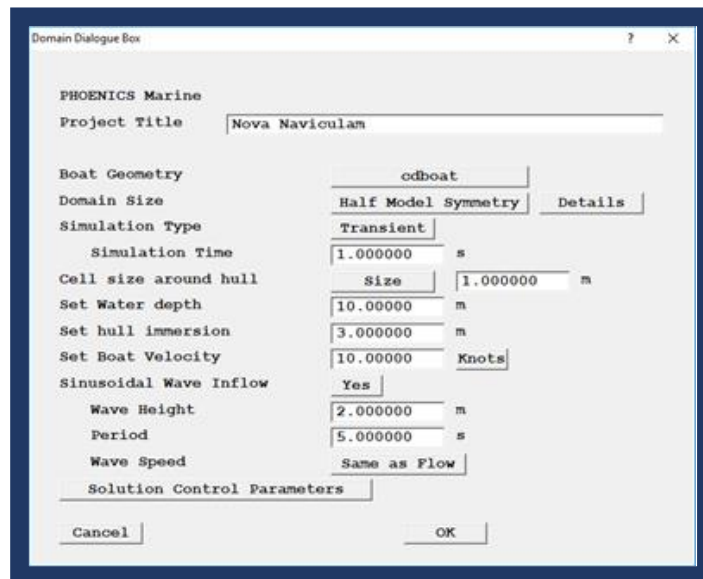
The Universal Thermal Climate Index UTCI provides an assessment of the outdoor thermal environment in bio-meteorological applications based on the equivalence of the dynamic physiological response predicted by a model of human thermoregulation, which is coupled with a state-of-the-art clothing model.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

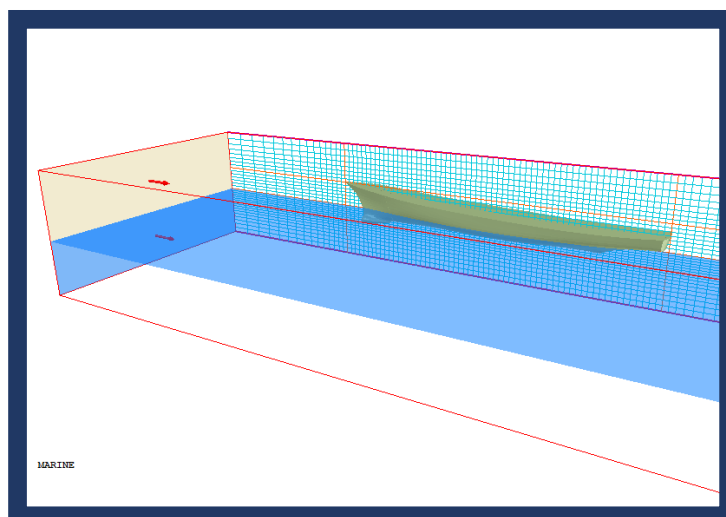
The published subroutine returns the UTCI as a function of air temperature, water vapour pressure, mean radiant temperature and wind speed 10m above ground level. In the Flair implementation, the local air temperature is taken to be the solved temperature TEM1, the local water vapour pressure is derived 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. This is achieved via a simplified menu interface dedicated to this type of simulation.



Employing the free-surface options already available in PHOENICS, the PHOENICS Marine SPP assists users to import or specify their particular hull shape and key parameters of their simulation - such as flow velocity and waterline location - whilst automating aspects such as domain generation, mesh definition and selection of appropriate relaxation parameters.



PHOENICS Marine will return values for drag, separated into skin friction and form drag while displaying and quantifying the size of bow and stern waves produced. An added innovation allows users to test their designs with sinusoidal waves as an input - see page 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 that will allow the hull to react dynamically to flow, enabling the trim calculations of interest to naval designers.

The Marine SPP will also be activated in CHAM's CFD "Plug In" to McNeel Corporation's Rhino3d CAD software. It will be known as *RhinoCFD Marine* and should greatly benefit the large number of naval engineers who already use this package.

Directory Structure

The alternative pre-processor, Prelude, has been withdrawn from the standard delivery. As a result, the DATMAKER utility used to read CAD files has been moved into the new `\phoenics\d_utils\datmaker` folder.

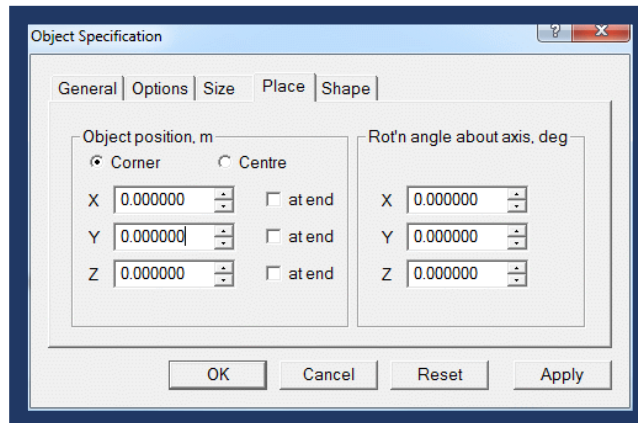
The 'textures' folder has moved to `\phoenics\d_satell`.

Items in PHOENICS 2017 added since the last release of PHOENICS 2015 (dated October 2016)

VR-Editor

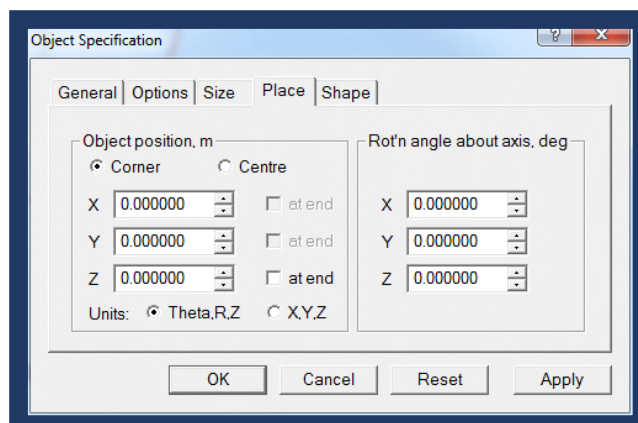
New options for object position

In Cartesian co-ordinates, the position can define:



- the location of the corner of the bounding box nearest the origin (the default, as before), or
- the location of the mid-point of the bounding box.

In Polar co-ordinates, the position can define the:

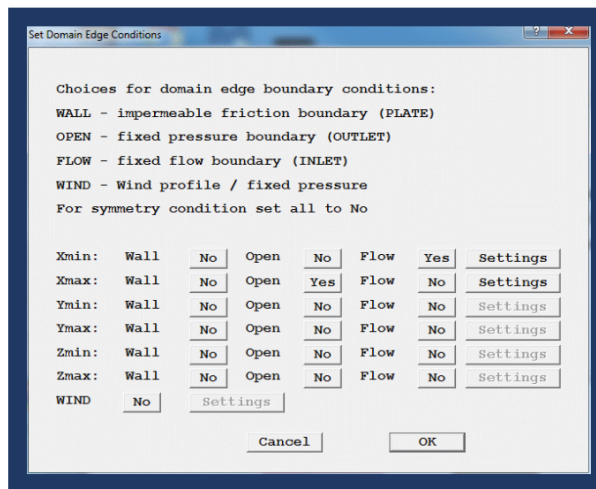


- location of the corner of the bounding box nearest the origin in (θ,R,Z) (default, as before), or
- location of the mid-point of the bounding box in (θ,R,Z) , or
- location of the corner of the bounding box nearest the origin in (X,Y,Z) ; or
- location of the mid-point of the bounding box in (X,Y,Z) .

When the position is set to 'Centre' and the Size is changed, the object will expand sideways on both sides, leaving the centre position unchanged.

The new options make it very easy to create a new object centred on an existing object, or in polar co-ordinates to locate an object symmetrically over the domain Z axis.

Updated Domain Edge Condition Dialog

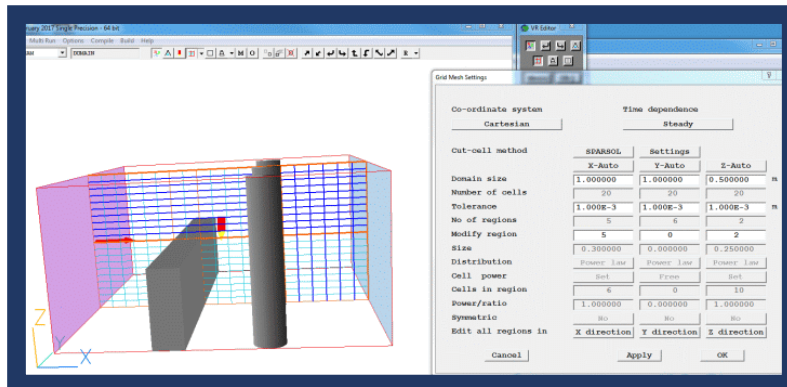


The new 'Settings' button on each line opens the Attributes dialog of the Inlet / Outlet / Plate object created for that face.

If the WIND is activated, any face objects are deleted and a WIND object is created. 'Settings' opens the WIND attributes dialog.

Highlighting the Selected Grid Region

When adjusting the grid by region, the currently-selected region is highlighted in the main graphics window.



This makes it much easier to determine exactly which region is to be modified.

POINT_HISTORY Object

The POINT_HISTORY object has been updated to allow it to work for steady-state cases, as well as for transients. In addition, the frequency with which the data is written can be set. This allows users to easily create multiple monitoring points to assess convergence behaviour.

Each POINT_HISTORY object will create a (number of) .csv file(s) showing how the variables selected for monitoring change with sweep or time at that point.

Each .csv file can contain up to 6 monitored variables, so if for example, 8 variables (say P1, U1, V1, W1, KE, EP, TEM1, SMOK) are monitored, two .csv files will be created. The first will hold P1, U1, V1, W1, KE and EP. TEM1 and SMOK will be in the second.

Selecting the object in Viewer, then selecting 'Show results' from the right-click context menu will display the data in graphical form. The Viewer knows how to assemble the data from the individual .csv files.

The .csv files can also be loaded directly into Excel.

Q1 – PIL

New PIL Commands

New PIL command INTPOL to perform linear interpolation in a table:

```
INTPOL(file_name, Xcol, Ycol, Xvalue)
```

The file can contain any number of named columns. The command is also available as an InForm function – an example is given below under '[New InForm Command](#)'.

The Inform PWLF function is implemented as a PIL command. This also performs linear interpolation in a file, but the file can only contain two columns. The first is always the X value, the second always the Y value.

READARR Command

The READARR command is used to read the contents of a data file into a PIL array. Previously, this was limited to 2D arrays, but now has been extended to also handle 3D arrays.

VR-Viewer

General Improvements and Corrections

When a ROTOR object is present and the true X-direction velocities TRU1 and TRU2 for ONEPHS=F are STORED, the vectors automatically use these as the X-component. Vectors drawn using U1 are meaningless, as U1 has different frames of reference inside and outside the rotor.

Corrections to plotting Fine Grid Volumes:

- Limits of FGV used to obtain cell-corner values reduced by 1 cell in each direction to eliminate artefacts at FGV edges.
- Vector plotting at edges of FGV corrected.
- Skip cells covered by another block when evaluating minimum and maximum values.

Solver – Earth

New and Upgraded Turbulence Models

- **High-Re Realisable k- ϵ model**

The realisable k- ϵ model has been introduced into PHOENICS. This model is popular in the CFD community due to its improved performance over the standard k- ϵ model when applied to flows involving boundary layers in strong adverse pressure gradients, streamwise curvature, separation and recirculation zones. The model is also reported to improve significantly the predicted spreading rates of round jets. The model is a two-equation high-Reynolds-number turbulence model that differs from the standard k- ϵ model in two respects. First, the model employs a different formulation of the transport equation for the dissipation rate that is derived from the transport equation for the mean-square vorticity fluctuations. Secondly, the model uses a different eddy-viscosity formulation which is based on several realisability constraints for the turbulent Reynolds stresses. In practice this means that the eddy-viscosity coefficient C_μ is a function of local flow parameters, rather than a constant, as in the standard k- ϵ model.

The model can be activated from the VR Main Menu / Models panel, and is compatible with the WIND object, scalable wall functions and fully-rough boundary layers with finite displacement heights.

- **Extended High-Re Wilcox-Kolmogorov k- ω model**

This model has been extended and tested (via library cases) to handle fully-rough boundary layers

(GRND5), with finite displacement heights (WALLB>0). The WIND object has also been extended and tested for use with the k- ω model.

The model can be used with scalable wall functions.

- **Scalable wall functions**

Scalable wall functions ensure that the wall distance employed in the wall functions is such that $y^+ \geq 11.126$ irrespective of the level of near-wall grid refinement. This value of y^+ marks the intersection of the linear and logarithmic velocity profiles. It is evident that the scalable wall function will produce identical results to the standard wall function for values of $y^+ \geq 11.126$.

Scalable wall functions can be activated for all high-Reynolds-number turbulence models that employ equilibrium wall functions without surface roughness. They are not currently implemented for use with non-equilibrium wall functions or with the Reynolds Stress Transport Model.

GENTRA

New option to read the droplet/particle drag law curve from a data file. This is used in the RAIN model. It is activated by setting RG(97)=GRND2, or from the GENTRA menu.

New InForm Function

New InForm function to perform linear interpolation in a file.

INTPOL(file_name, Xcol, Ycol, Xvalue)

This is an enhanced version of PWLF, in that the file can contain any number of named columns. Each file is read only once at the start of the run. Multiple references to the same file will pick up stored values already read in – a file will only be read and stored once no matter how many times it is referred to.

For computational efficiency the interpolation uses a binary search to find the correct segment of the table given the input Xvalue. The differences in the table are also computed once and stored.

As an example, let file ethglyc.csv contain the values of enthalpy (ENT), specific volume (SPV), dynamic viscosity (EMU), conductivity (CND) and diffusion coefficient (DIF) as functions of temperature (TEM):

TEM,	ENT,	SPV,	EMU,	CND,	DIF
-17.8	, -1238746.805,	0.000862069,	0.000115882,	0.24813444	, 9.46135E-08
4.4	, -1171705.951,	0.000873362,	5.85505E-05,	0.252724	, 9.38042E-08
26.7	, -1104278.562,	0.000884956,	1.92630E-05,	0.256915667,	9.20403E-08
48.9	, -1037069.369,	0.000896861,	6.34036E-06,	0.259949667,	9.09873E-08
71.1	, -969776.1961,	0.000909091,	3.90773E-06,	0.261084667,	8.85781E-08
93.3	, -902399.0434,	0.000922509,	2.28581E-06,	0.260923667,	8.7108E-08
115.6	, -834633.8417,	0.000937207,	1.43921E-06,	0.259341333,	8.4257E-08
137.8	, -767088.3508,	0.000952381,	9.49333E-07,	0.255937333,	8.19981E-08
170	, -668967.72	, 0.00100038	, 6.55049E-07,	0.2475	, 8.11784E-08
197.5	, -585029.1675,	0.001025599,	4.84083E-07,	0.2366	, 7.60679E-08

To use this file to set the kinematic viscosity (VISL = dynamic viscosity * specific volume) at the local temperature (TEM1), one could use:

(PROPERTY VISL IS INTPOL(ethglyc.csv,TEM,EMU,TEM1)*INTPOL(ethglyc.csv,SPV,TEM,TEM1))

Updated InForm TABLE Command

The InForm TABLE command produces a file containing tabulated values as a function of sweep (last argument !SWEEP) or time (last argument !TIME). These values have been written every sweep or every time step, which can result in very large files.

It is now possible to add a further argument specifying the frequency of writing. For example:

- (TABLE in file_name is GET(variables) with HEAD(column_headers)!sweep!ifreq) or
- (TABLE in file_name is GET(variables) with HEAD(column_headers)!time!ifreq)

will write to the named file every ifreq sweeps or steps.

General Improvements and Corrections

Corrections to Fine Grid Volumes:

- Skip patches not in current domain.
- Correction to monitor plot to avoid 'spikes'.
- Ensure loops don't go out of bounds in FGV linear equation solver.

Issue 'Please wait' message when saving dump files, so that user is aware something is going on.

Change the monitor plot of Maximum Correction to show log10 of maximum correction. This makes it easier to see small changes later in the solution without them being swamped by big changes early on.

The image on the left shows the old linear plot, the one on the right the new log10 plot.



Correction to printing the sum of InForm sources to RESULT. If there is a whole-domain InForm source, but no source is actually found at IZ=1 due to the IF() conditions, the printed source sum is never reset to zero and just grows with sweep and step. (The source applied is correct).

In MOFOR if the ROOT in the MOF file was given the name 'root', the numbering of the frames of reference, and hence OBID numbers, was out by one.

Corrections to Sparsol detection in Polar co-ordinates.

The printout of inter-phase transfer sources was missing in parallel. The sources were active and correct, just not echoed to RESULT. This is fixed.

Corrections to ROTOR objects defined by CAD shapes.

Stuck velocity residual when solving velocities whole-field in IPSA is fixed.

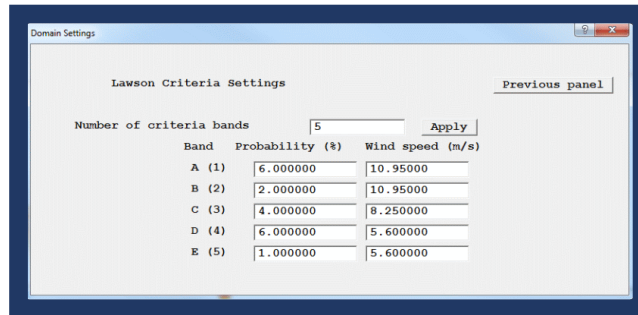
Obscure crash when blockage object is called MATERIAL is fixed.

If the license file expires during a run, allow the run to save the output files if the license file is renewed while Earth is paused.

Flair / Flair-EFS

Lawson Criteria for Pedestrian Wind Comfort

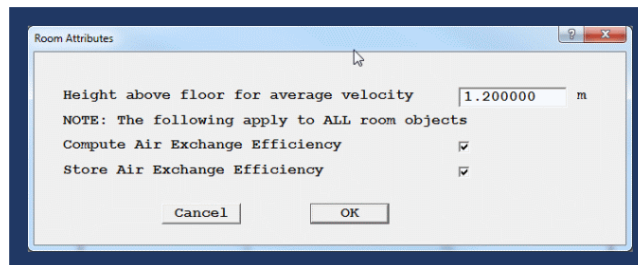
The Lawson Comfort Criteria define a range of pedestrian activities, and for each activity define a wind speed and maximum frequency of exceedance. If the wind speed exceeds the threshold for the activity the conditions are unacceptable. A default set of criteria have been implemented, together with means of inputting different probability and threshold velocity values.



The PHISUM utility, used to sum results over computed wind sectors, has been updated to re-compute the Lawson Criterion (LAWS) based on the summed probabilities of the individual bands.

Air Exchange Effectiveness

The ROOM_ACH object has been renamed as ROOM.



The Air Exchange Effectiveness (AEE) for each ROOM can be evaluated. A variable called AGE2 is solved, and is given a uniform source of 1 per-unit-mass throughout each ROOM. At the entrance to each ROOM it is set to zero. The AEE is then calculated as the sum of all outflow AGE2 from a ROOM divided by the average AGE2 in the ROOM.

AGE2 is the equivalent of the Mean Age of Air (AGE), but for each individual ROOM.

In addition, the ROOM object also calculates and prints:

- The total volumetric inflow rate in m³/s and m³/hr, including flows across the boundaries of the ROOM and any internal sinks;
- The volume-weighted average temperature within the ROOM;
- The minimum and maximum temperatures within the ROOM;
- The area-averaged absolute velocity at a given height above the floor of the ROOM. Note that the floor is always taken to be the lowest Z plane within the ROOM.

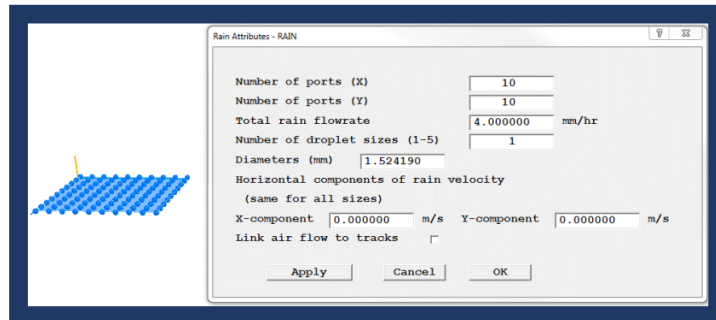
A typical output is shown here:

```
For object ROOM1
-----
Overall residence time calculated as
free volume/volum.flow-rate (in seconds)
RES.TIME=6.731647
Ventilation rate in air changes per hour
ACH =534.787476
The total free volume in the room is (m^3)
VOLUME =1639.037354
The total volumetric flow rate is (m^3/s, m^3/hr)
VFLRT s =243.482391 ;VFLRT h=8.765366E+05
```

The total internal volumetric source is (m^3/s , m^3/hr)
VFLRT s =20.00001 ;VFLRT h=7.200003E+04
The volume-weighted average temperature
Ave Temp=21.344286
The minimum and maximum temperatures
Min Temp=20.013987 ;Max tem=26.08449
The Air Exchange Effectiveness
AEE=0.991637
The area-averaged absolute velocity at 1.2m
VAB_AVE=4.915433

Rain Object Update

The horizontal components of rain velocity can be set.



Each ball represents a GENTRA droplet injection port. Each port can emit up to five droplets of different diameters.

General Improvements and Corrections

The default value of the maximum increase allowed for KE (MAXINC(KE)) has been increased to 100 to prevent hitherto unexplained sudden divergence in some parallel cases – which run perfectly in sequential mode.

ESTER

Optional new formats for the magnetic field file, MAGF, cathode currents file, JZC and anode height file HANO have been introduced. These allow the grid implied by the data file to be different from the actual grid used in the solution. The magnetic fields and cathode currents are interpolated onto the solution grid, allowing the same input file to be used for a number of meshes.

Directory Structure

Up to now, the executables for PHOENICS executables have always been in the folders

```
\phoenics\d_satell\d_windf\satexe.exe
\phoenics_d_earth\d_windf\earexe.exe
```

and so on for other modules. This has made it impossible to install 32-bit and 64-bit versions simultaneously. The directory structure has been changed to:

```
\phoenics\d_satell\d_win32\satexe.exe
\phoenics\d_satell\d_win64\satexe.exe
\phoenics_d_earth\d_win32\earexe.exe
\phoenics_d_earth\d_win64\earexe.exe
```

This makes it possible for 32-bit and 64-bit versions to co-exist.

Items in PHOENICS 2015 added since the last release of PHOENICS 2015 (dated 31 March 2016)

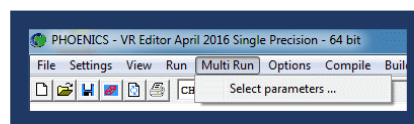
VR-Editor

Parameterised Multi-run

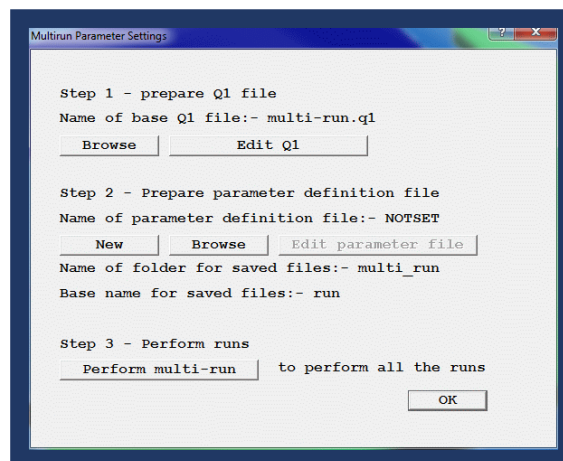
The multi-run facility allows several runs to be made in sequence, using different values for certain parameters in each run. An example might be to simulate the flow around some buildings with different wind directions and/or speeds. The process consists of four main steps:

- Create the 'base' model, and save it 'as a case'
- Edit the 'base' Q1 and replace values with parameters. Each parameter name starts and ends with @, e.g. @param1@, @param2@. The user can insert as many parameters as required. Each parameter must evaluate to a setting the pre-processor will recognise.
- Create or modify a parameter definition file, which contains the names of the parameters, and the value to use for each run.
- Execute the multi-run sequence. For each run specified in the parameter file, the parameters in the base case will be replaced by the values in the parameter file, the Pre-processor and Solver will be executed and the results saved.

The Multi-Run menu consists of the following item:



Once the base case has been identified, the main dialog used to set up and perform parameterised multi-runs will open. This dialog first appears as:



Linked ANGLED-IN objects

A user commented that although he liked the ability to add a source of heat to linked ANGLED-IN objects, he would really like to set the exit temperature. This has been implemented, together with an option to set the temperature rise. The heat source required to maintain the exit temperature or temperature rise can be determined from RESULT.

General Improvements and Corrections

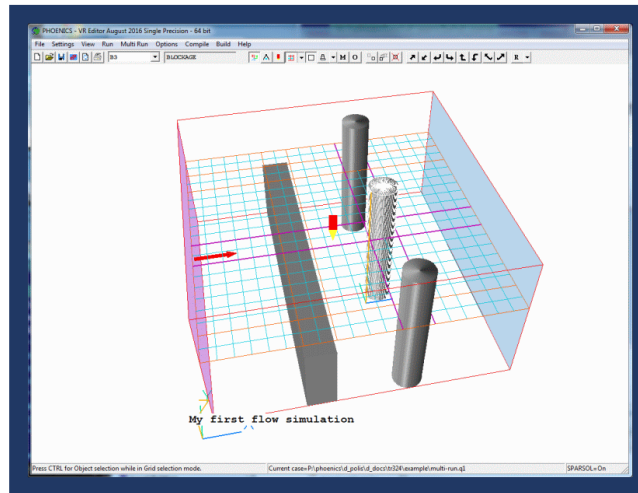
On the INLET dialog, the current density is displayed when inlet density is set to 'Domain material'.

It has only been possible to use a domain-material BLOCKAGE to set the initial distribution of selected variables and scalars. It has been made possible to set initial values for any stored or solved scalar using a BLOCKAGE object.

A user found that the fan-matching option did not work well when the fan was on the domain boundary and set to extract air from the domain. The dialog for setting the fan direction was also not clear. The FAN dialog has been reworked so that when matching is active, the dialog only asks for a positive or negative direction, not a positive or negative value. The way fan-matching works for swirl fans has also been improved.

An item has been added to the right-click object context menu 'Move to object centre'. Previously the menu contained 'View centre' and 'Probe position'. The new option combines these, allowing the view centre and probe to be moved to the centre of the selected object in one step.

Regions created by an object are highlighted in purple in the mesh display when the object is selected.



In polar coordinates, the X position of an object or the probe is restricted to lie between 0 - 2π radians. When passing 2π interactively, the location is reset to 0.0.

Q1 – PIL

A new PIL variable NOCLIP has been introduced. When T, all objects are set to be not constrained by the domain when the Q1 is first processed. When F, it has no effect.

VR-Viewer

General Improvements and Corrections

Correction to VR-Viewer when finding sources in RESULT for an object.

When VRV writes a file containing surface contour coordinates and values, the file contains many duplicate lines and hence is far too big. A method has been implemented for testing each line before it is written to see if it has been written before, and skipping it if it has. This has greatly reduced the file size, at the cost of a delay in writing.

The probe display can be changed from a 'pencil' to a sphere. The colour and diameter of the sphere can be changed. The probe appearance is saved to and read from the macro.

Objects can display surface contours and vectors simultaneously. Previously they could show one or the other.

Solver – Earth

GENTRA Speedup

Coding in the GENTRA parts of the Earth solver has been reviewed to improve efficiency. Several changes to coding practices have led to an approximately 3 - 4 times reduction in CPU time for GENTRA. As an example, a 50 sweep run with 30,000 GENTRA tracks took 480s to complete; it now takes 120s.

Solver Volumetric Block-Corrections

The default Stone-type linear equation solver used in Earth has long been able to apply block-corrections over specified volumes. The draw-back has been that the blocks had to align with the grid, making it hard to fit irregular shapes. The Volumetric Block Corrections have been re-written in a more flexible fashion, allowing for easier definition of the blocks. This is further enhanced by the new ability of the Editor to set initial values of any scalar, including BLOK, at a BLOCKAGE object.

General Improvements and Corrections

It was noticed that if a run continues for very (very, very) long it could become impossible to interact with GXMONI. The time taken to refresh the screen is so long that the interrupt-screen does not appear correctly. This has been fixed by changing the way monitor plot lines are drawn. Rather than drawing them as individual segments, each line is now drawn as a single poly-line, which is much more efficient.

In two-phase cases, if GEN2 is present the first-phase generation terms and gradient stores are contaminated by second-phase values, and second-phase values are also incorrect. This was caused by use of the same temporary stores for both phases. Storage of second-phase velocity gradients DU2X - DW2Z, which were not obtainable, has been provided. The names have been added to the reserved list.

Recent Consultancy work using ASM has highlighted some omissions, including:

- Display of PTO residuals on the graphical convergence monitor screen and in result;
- Reporting of particle nett sources;
- Setting mass sources with InForm when ASLP is active

Corrections to InForm in parallel. For long formulae, the link between an object/patch and command could be lost. If the object/patch was not on all processors, they could get out of step and hang.

Corrections to InForm to prevent internal confusion between an object called TABLE and the InForm (TABLE in ...) command.

If the scalar wall function is entered with a zero relative velocity, NaNs and Inf are produced. This can happen in Sparsol for a cut link. This is fixed.

It has been reported that for very complex geometry on a very fine grid, the geometry detection appears to go wrong after a certain Z plane. This is traced to the way the IJK indices of the cut links were being stored. Eventually, accuracy of the index was lost due to round-off error. This is now fixed.

InForm sources can have a 'with whol' or 'with volu' tag. When the source is applied to a faceted object, the two tags require different patches. The checking in Satellite and Earth to ensure that a suitable patch was always used has been tightened up.

Some ROTOR cases can crash when exiting Earth, if they are set to calculate the integrated pressure force on the blockages. The causes of this have been traced and fixed.

In two-phase cases run in parallel, the interphase transfer rates were not echoed to RESULT.

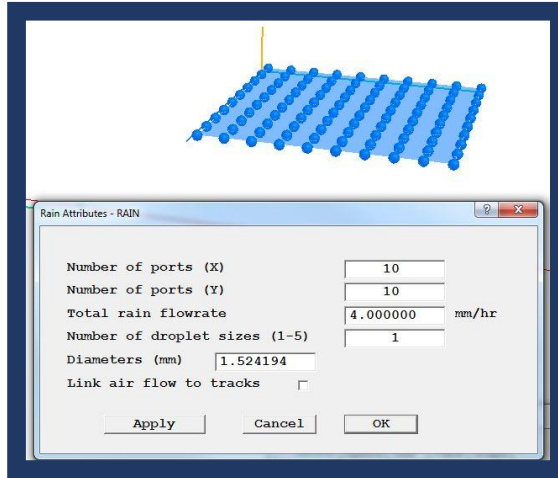
In polar coordinates, objects which straddle the $X=0$, $X=2\pi$ boundary did not produce the correct source or PRPS distribution as they were not detected fully. This is fixed.

Correction to MOFOR in polar coordinates. There were spurious polar-cartesian transforms in one routine, and missing transforms in another.

Flair / Flair-EFS

New RAIN object for Flair

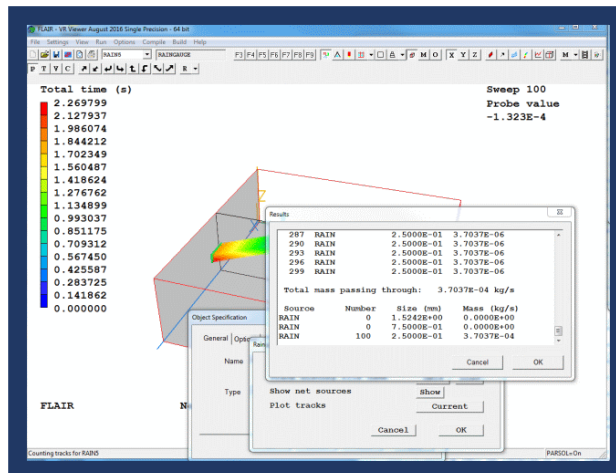
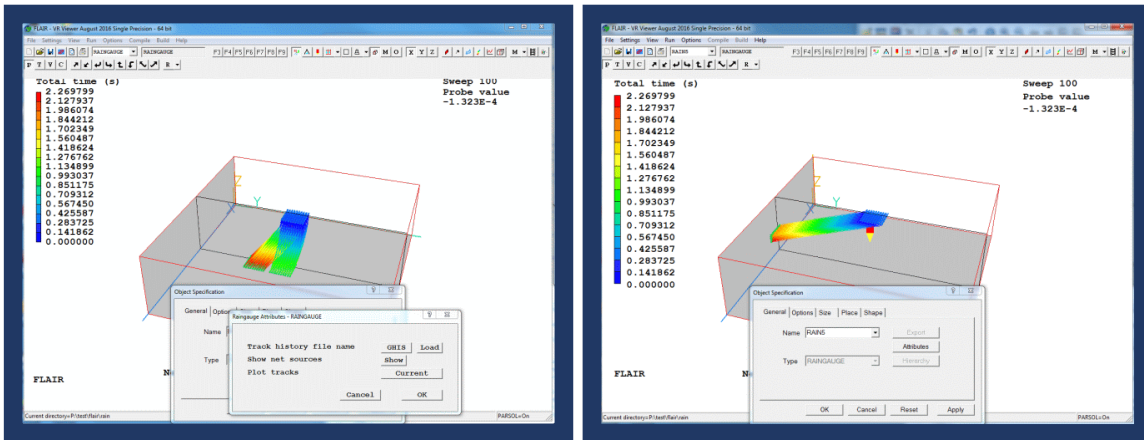
The RAIN object utilises the GENTRA module to track the paths of clusters of rain drops of differing sizes.



Rain tracks are initiated from a regular grid of 'ports'. Up to five different droplet sizes can be specified. The total mass flowrate of rain is evenly divided amongst all the tracks. The onerous task of creating the GENTRA inlet table file is automated and hidden from the user.

New RAINGAUGE object for Flair

The RAINGAUGE object provides a means of showing where individual tracks go, and summing up the mass of water passing through a surface.



The RAINGAUGE object also performs the same function for GENTRA tracks originating from the SPRAY-

HEAD object. It has limited functionality for manually-created GENTRA tracks, but this is a possible area for future development.

General Improvements and Corrections

The default number of iterations for Flair-EFS has been increased from 100 to 2000.

When a spray-head object is introduced in Flair, GENTRA is activated, the water-vapour equation MH2O is turned on and the density is set to 'Ideal Gas, Mixture Molecular Weight'. This density option relies on the scalar being C1 (variable 16), and it was possible for MH2O to appear in some other variable. This led to crazy densities. It has been ensured that however MH2O is activated, it will appear in variable 16.

Cases using Flair and GENTRA together can crash if DIFFUSER objects are used. When GENTRA is active, mass-flow patches are renamed to start 'GX' in order to act as exit boundaries for the particles. This was not done consistently for the diffuser leading to a run-time error-stop. The problem is fixed.

Items in PHOENICS 2015 added since the last release of PHOENICS 2015 (dated 26 Oct 2015)

VR-Editor

General Improvements and Corrections

There was an error reading the near-plane position from Q1. The effect is that cases with small dimensions appear as a blank screen until 'Fit to window' has been clicked. This is fixed.

There was an error in the formula used to set the initial EP profile for the WIND object when the inlet profiles are read from a table. The initial EP should be $CD/AK*(PWLF(file,ZG)^{1.5})/ZG$ not $CD/AK*PWLF(file,ZG^{1.5})/ZG$. The missing () are added.

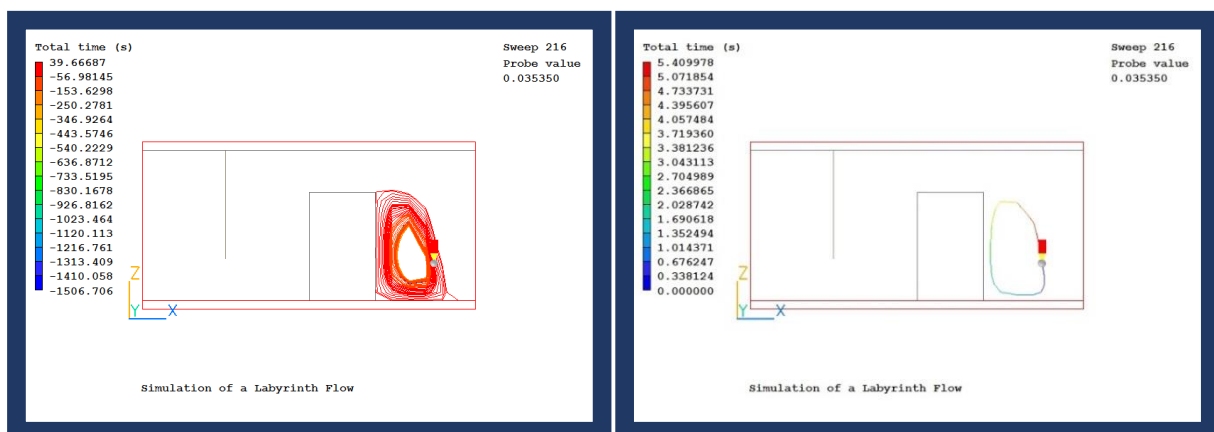
The rotation centre was not initialised correctly, leading to strange behaviour when newly-created objects were added to an assembly object. This is fixed.

ASSEMBLY objects are created in wireframe mode by default, allowing the contents to be seen.

VR-Viewer

Plotting Closed-loop Streamlines

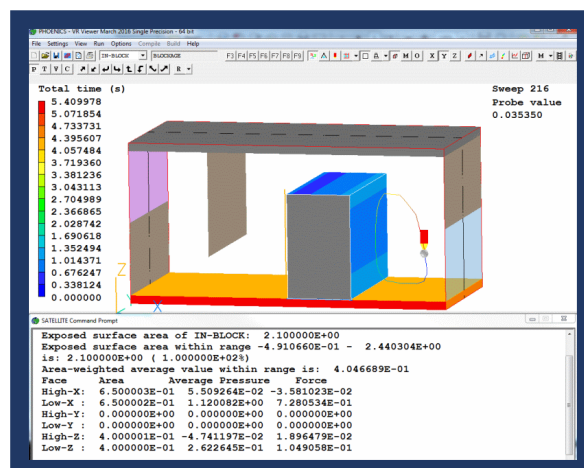
If a streamline passes through the cell containing the start-point of the track, and the track then passes with half a cell size of the start-point, the track is terminated and drawn as a closed loop.



The image on the left shows the previous streamline plot, the on the right the new closed form.

Average Values

The area-weighted average value of a variable over a surface can be obtained by turning on 'Surface contours' for any object, and then 'Dump surface values'. The next image shows a typical output:



General Improvements and Corrections

Corrections to do with how often surface contours are regenerated, to prevent unnecessary contour regeneration when moving the probe.

The VRV macro command to plot a line plot caused the execution of an infinite loop, making VRV freeze. Further, the macro command to dump an image file of a line plot did nothing. These are fixed.

Solver – Earth

Solution of Scalars in Solids

Normally, the solution and display of all scalars (except TEM1, TEM2, H1, H2, EPOT and STOREd scalars representing physical properties) is suppressed inside solids. In addition, it has been possible to display derived variables inside solids if the NAME of the variable started with the '#' character, e.g. #C1. It is now possible to also solve for such variables.

The VR-Editor has been upgraded to allow sources for such scalar variables to be set at participating blockages and at plates.

Reporting of KE and EP Sources

The source of KE and EP in the two-equation turbulence model is applied through the KESOURCE patch. For the default linearization, KELIN=3, the source is split into linearised and non-linearised parts. Only the linearised part has been reported in the 'Sources and Sinks' section of RESULT, making it hard to judge the degree of source balance. The non-linearised part is now also reported.

Auto-creation of CSV files from PROFIL patches

A PATCH command with type PROFIL can be used to create a table of values and a line-printer plot in RESULT. The table of values is now automatically written to a CSV file at the end of the run, making plotting profiles in Autoplot or Excel easier. For time history profiles, the X- axis is now the actual time at the end of each time step not the step number.

In parallel, data are written by the processor having the patch. Distance plots which straddle processor boundaries will be written on each processor they cross. Time histories will also appear on the processor the patch is located in.

For distance plots in transient cases a separate file is written each print step. The name has the step number after the patch name e.g. prof000101.csv.

If the patch is linked to a VR object, e.g. a POINT_HISTORY object, the object name is used as the base of the file name since the patch name will be meaningless to the user.

Corrections to Scalar Equation Model (SEM)

In the Scalar Equation Model, convection terms of the SURN equation do not contain density. Erroneously, the convection terms of other SOLVEd variables also did not include density. This is fixed.

Improved reporting of residuals of SURN in the Scalar Equation Model.

General Improvements and Corrections

If the RESULT file is being held open in another program when the Earth solver tries to write to it, a warning is issued and the user is given the opportunity to exit the other program and try again. Previously the Earth run would crash.

Improvements to the 1-D block corrections in parallel to prevent occasional NaNs.

If a transient run is ended on a step which is not a printing step, the residual table written to RESULT contained 1E19 instead of the residuals for that step. This is corrected.

When a run fails it can be difficult to analyse the reason without a phi file. A phi file will always be written when a failure is detected. The name of the file will start #.

Add the ability to set diffusivity for velocity equations using InForm, for example (Property of PRNDTL(U1) is -1).

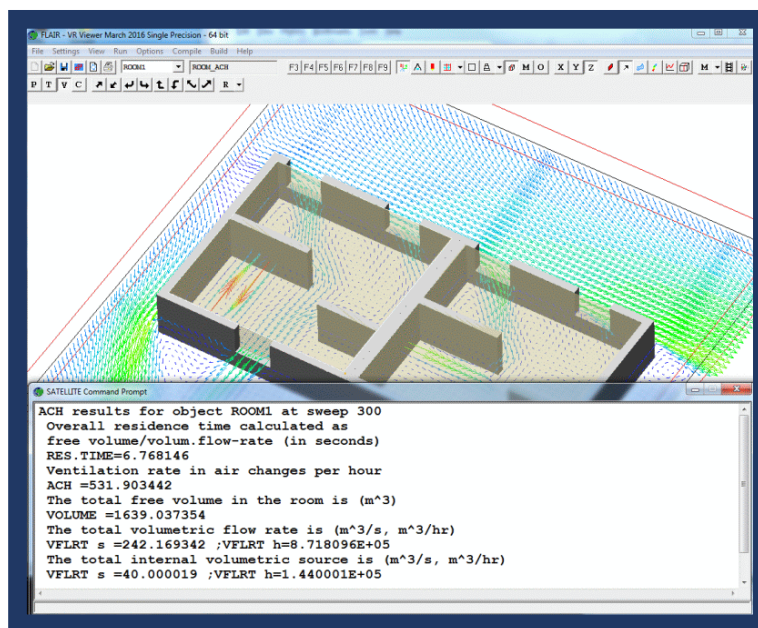
Some cases using PARSOL can take a (very) long time to start calculating. This has been traced to an inefficient algorithm for detecting cells cut by more than one object, and has been corrected.

Flair / Flair-EFS

Air Changes per Hour for a closed volume

A ROOM_ACH object type has been introduced. This defines a closed volume, for example a room within a building. The sum of the volumetric inflow rates through the faces of the object is computed, and is divided by the free volume within the object. This is then multiplied by 3600, to give the number of Air Changes per Hour for the object. The ACH value is printed to RESULT, and can be displayed in the Viewer by right-clicking a ROOM_ACH object and selecting 'Show net sources'.

The next image shows a typical usage for flow through a building:



Two ROOM_ACH objects are seen, one in each room. As the objects align with the room walls, the flow rates calculated are the inflows through the windows. The flow from the internal inlets is also captured.

Apparent Temperature – Heat Index

Two forms of Apparent Temperature have been added, based on a mathematical model of an adult, walking *outdoors*, in the shade. The version including the effects of temperature, humidity, and wind is:

$$AT = Ta + 0.33 \times e - 0.70 \times ws - 4.00$$

The version including the effects of temperature, humidity, wind, and solar radiation is:

$$AT = Ta + 0.348 \times e - 0.70 \times ws + 0.70 \times Q / (ws + 10) - 4.25$$

where:

Ta = Dry bulb temperature (°C) (PHOENICS TEM1)

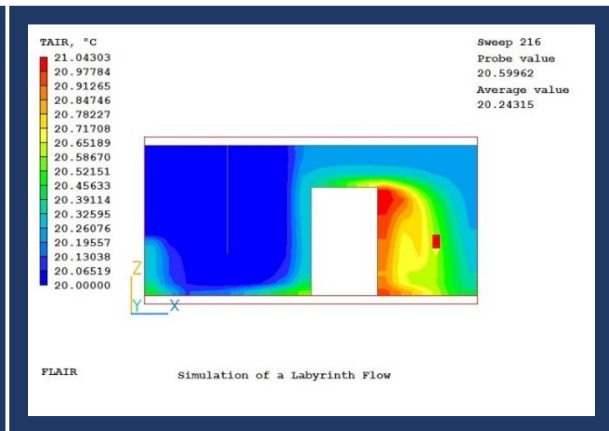
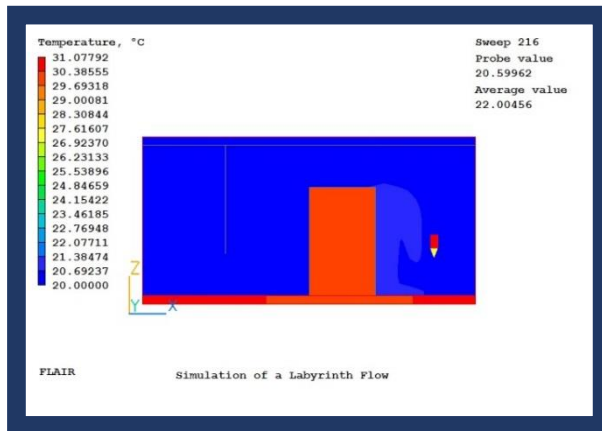
e = Water vapour pressure (hPa) [humidity] (PHOENICS PVAP)

ws = Wind speed (m/s) at an elevation of 10 meters (PHOENICS VABS)

Q = Net radiation absorbed per unit area of body surface (w/m²)

Air Temperature (not including Solids)

The variable TAIR has been added to the Comfort Index panel. This is just the temperature, TEM1, but only in cells containing fluid. The images below show the normal temperature, TEM1 on the left and the air temperature, TAIR, on the right:

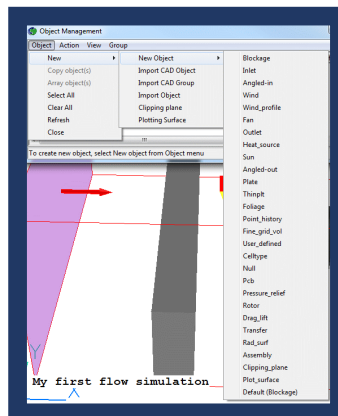


Items in PHOENICS 2015 added since the last release of PHOENICS 2014 (dated 19 Jan 2015)

VR-Editor

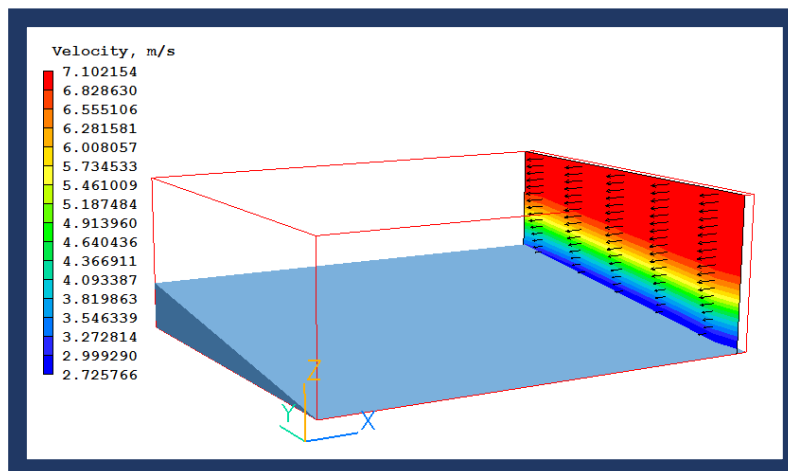
Object Creation

When a new object was created, the default object type was always BLOCKAGE, which often had to be changed to the required type. The Object / New / New Object menu now immediately offers a list of object types so that the right type can be selected right away.



WIND Object

The WIND object can now take inlet profiles of velocity and turbulent kinetic energy from files. The files contain tables of velocity or turbulent kinetic energy against height.



The image shows a tabular inlet profile entering the domain over a wedge representing the terrain.

The WIND object initial velocity log-law profiles are fully compatible with in-Earth inlet profiles by adding tests on cell half-width less than the roughness height and non-zero displacement thickness.

PLATE Object

In a transient case, the PLATE now operates fully within the set time limits – the porosity is also switched off (set to 1.0) outside the time limits e.g. to represent opening / closing door. Previously only the sources attached to the PLATE obeyed the time limits, the porosity was always present.

CAD Import

The maximum number of CAD files selectable in one go is now a CHAM.INI item, defaulted to 500. Should a user wish to import more than 500 CAD files in one selection, they need to change MAXCAD in the CHAM.INI file.

```
[Satellite]
MAXCAD = 500
```

The import of CAD files would fail if there was more than one '.' in the file name, e.g. 'part.1.3ds'. This is fixed.

Mesh Generator

The graphics screen is refreshed whenever 'Apply' is clicked in the grid menu. This makes it quicker and easier to see the effects of changing grid parameters.

Miscellaneous Corrections

Patches with no COVALs are deleted before writing the eardat file. This makes the eardat file smaller and reduces 'clutter' in the RESULT file echo.

The echoing of ANGLED_OUT objects to Q1 in two phase cases has been corrected. The phase-2 pressure coefficient was not written.

Allow for NX / NY / NZ greater than 999 when writing GCV multi-block link information.

When writing the Q1 for SEM echo heavy fluid at inlet /outlet as 'HEAVY', not 1.0.

The covals created at a SEM inlet are corrected to properly account for volume flow, mass flow or cartesian components.

Corrections to the linkage between object and patches for Inform.

Q1 - PIL

New PIL command:

```
SPLIT(arg1,arg2,arg3)
```

will take a character string **arg3**, split it into individual words using ',' as a separator and then place each word into elements of the PIL array **arg1**. **Arg2** returns the number of words found, and the dimension of the **arg1** array. If **arg1** does not exist, it is created.

New PIL command OBJINFO writes details about a selected object, or all objects, into a designated file.

The reading of the Q2 file is delayed until just before writing eardat so that all object information is present. This allows an OBJINFO command in Q2 to return details of all objects.

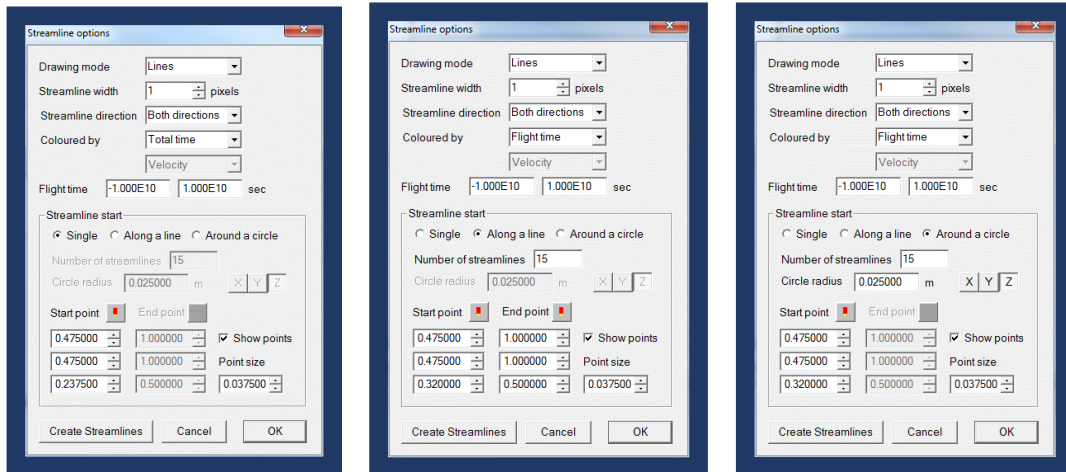
The use of environment variables in filenames in PIL WRITE and EXEC commands is allowed.

The handling of quote marks in PIL character strings has been corrected.

VR-Viewer

Streamlines

The dialog for setting streamline start positions has been updated:



The images show the new dialogs for starting streamlines at a point, along a line or around a circle. Everything required to specify the start location is on one dialog, and each time 'Create Streamlines' is clicked, a new set of streamlines is created. On clicking 'OK' to close the dialog, the existing 'Streamline Management Dialog' showing all current streamlines is displayed.

SPARSOL

Small corrections to SPARSOL plotting.

Add a search for minimum / maximum values in the cut-link section of the phi(da) file.

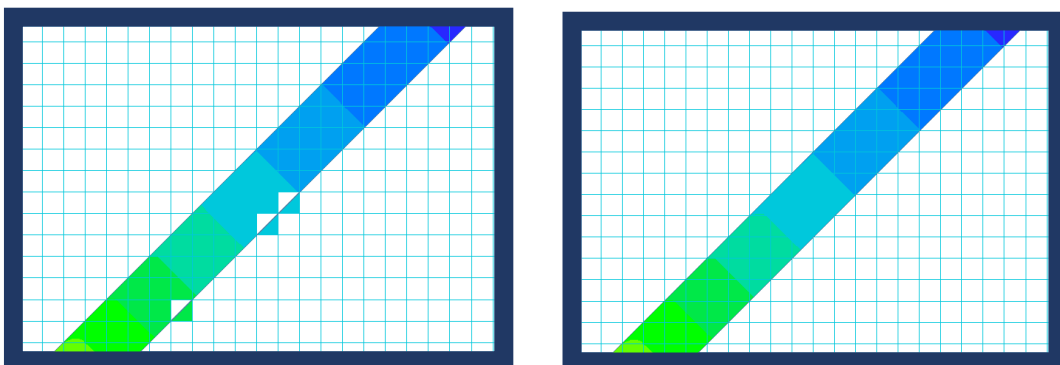
Don't try to read a pbcl.dat if one is present when SPARSOL active.

PARSOL

The tolerance used in PARSOL contour plotting is now a CHAM.INI item.

```
[Graphics]
PlotTol = 1.0E-7
```

In some cases when the facets pass exactly through cell corners, Viewer can get confused about which side of the facet is solid and which fluid:



The image on the left has too small a value of the tolerance. In the image on the right, the tolerance has been increased and the expected contour is seen.

Macro

File names longer than 20 characters can be used as input/output when extracting multiple data points.

An offset is allowed for the extraction location when extracting multiple data points. In this way, the extraction point coordinates can be relative to, say, a building origin, and the offset will convert to absolute coordinates relative to the origin.

Correction to reading the number of colours to be used from a macro.

General

Allow plot surface objects to be freely rotated.

Solver - Earth

ROTOR Object

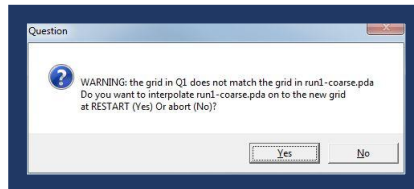
ROTOR objects can now overlap and /or touch each other. Previously they had to be separated by at least one cell in Y and Z. This allows more complex shapes to be built up from several primitives.

Corrections to ROTOR transient restart to ensure the correct initial values are used.

Corrections to the diffusion term at ROTOR boundaries to prevent occasional failure with NaN.

SPINTO-in-Earth

For many years the SPINTO (and earlier PINTO) utilities have been able to interpolate the solution fields from one grid to another. One use of this is to utilise a coarse-grid solution as the initial field for a fine-grid run. The option has not been used much as it required hard-to-create input files (Q1SPIN, Q1PIN) to control the interpolation. The interpolation is now handled within the Earth solver, and no extra commands are needed. When Earth sees that the grid on the restart file does not match the grid in the Q1, it shows this dialog:



If 'Yes' is clicked, the interpolation is performed and the run continues as normal. The method works for phida and phi files, in serial and in parallel.

General Improvements and Corrections

Corrections to the shock corrections applied when UCONV=T and NAMGRD=CONV. The effect of these corrections is to make the convective fluxes used in the momentum equations consistent with those used in the continuity equation. Under some circumstances temporary storage used to evaluate these corrections was overwritten in the solver before being used leading to wrong solutions.

Corrections to the transient term on first step of restart to avoid huge time fluxes. If physical properties were relaxed, the thermal capacity stored as old mass was based on a relaxed value, not the true value.

Capture source value from FIXVAL cells so that a sensible source value can be reported in the 'Nett Sources' section of RESULT.

Correction to the implementation of a fixed flux source in a PARSOL cut cell.

A summary of nett sources is printed if there are more than 10 patches

Sub-grid-size sources will still be applied (e.g. tiny heat source) in the cell(s) they are in, but blockage effects are ignored.

Additional InForm print frequency control NINFOR. The INFROUT file will be written every NINFOR sweeps as well as at normal print sweeps.

Corrections to the linear equation solver 1-D block correction sequence when solving H1, T3 or EPOT in conjugate situation where sources (and/or fixed values) are in the solid cells.

In first and last cells adjacent to blockage only a half-cell source is added, so in these cells divide SHR_X/Y/Z by 2 to ensure that the sum of SHR_X*cell area matches nett source and friction force printed to result.

Correction to prevent crash on Linux when saving image of final gxmoni plot.

Correction to SPARSOL cell-to-object linkage. If the object is above the cut link, the first cell is that where

the cell centre is above the intersection. If the object is below, the last cell is that with the cell centre below the intersection. This will ignore cells whose cell centres lie outside the facets, just as PARSOL did.

Use the same SPARSOL tolerance for sequential and parallel to ensure same detection. In parallel the relative tolerance was based on the local domain size, not the global domain size.

Allow GCV to store U/V/W1SL, U/V/W1AD, DP1X/Y/Z. U1SL are the post-solver UC1s, U1AD is the correction to U1.

Allow GCV to store PCOR.

Prevent a split in X direction for parallel if XCYCLE is used.

Issue an error stop for ASM in parallel.

In the SUN object allow flow boundaries and selected blockage/plate objects to be transparent.

Optional new termination criterion - set ENDIT(phi) negative and the run will stop when the maximum absolute correction for phi is below abs(ENDIT(phi)). For example, if ENDIT(P1)=-1, the run will stop when the biggest absolute pressure correction is less than 1.0.

Print the absolute maximum corrections (and their locations) to RESULT when printing residuals.

Correction to InForm CORR() operator. This only worked correctly for variables solved whole-field. It returned the wrong values for slab-wise variables.

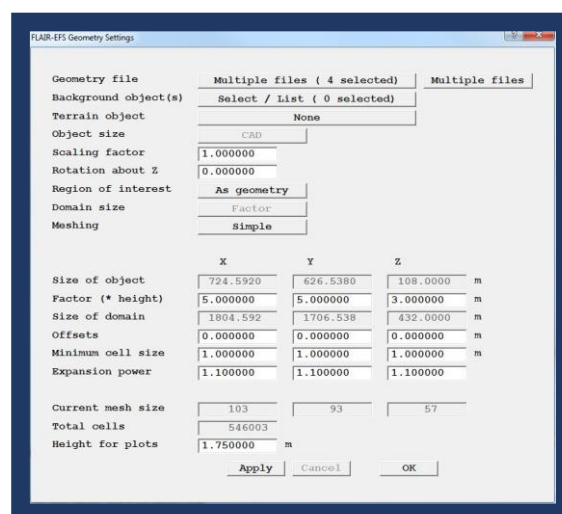
Correction to fixed-volume extraction applied to faceted shape. Source was multiplied by total object area instead of being divided, resulting in a source which was object-area-squared times too big (or small).

Flair-EFS

FLAIR-EFS is a reduced-cost subset of FLAIR for users concerned only with the external environmental conditions surrounding buildings and other structures. It is described [here](#).

The object types concerned with internal flows (such as jet fans, diffusers, spray heads) or fire modelling (fire) have been removed.

The normal mesh generator is replaced by a system based on the size of the building(s) being modelled, which are always placed centrally in the domain. The domain size is based on multiples of the maximum building height, and the user sets a minimum cell size. The FLAIR-EFS meshing dialog is shown below:



FLAIR-EFS can show Pedestrian Comfort information including:

- Wind Amplification Factors.
- Probability of the wind speed to exceed a set threshold value.
- NEN8100 Dutch standard for Wind Comfort and Wind Danger in the Built Environment.

Given a table of wind direction probabilities and wind speed probabilities for each direction, it is possible to create an average wind speed. This is done by running for each wind direction in the table, and then summing the absolute velocities multiplied by the sector probabilities. The PHISUM utility can be used to sum the output files from the individual runs.

Flair

Pedestrian Wind Comfort, also in Flair-EFS.

Averaged sector velocity, also in FLAIR-EFS.

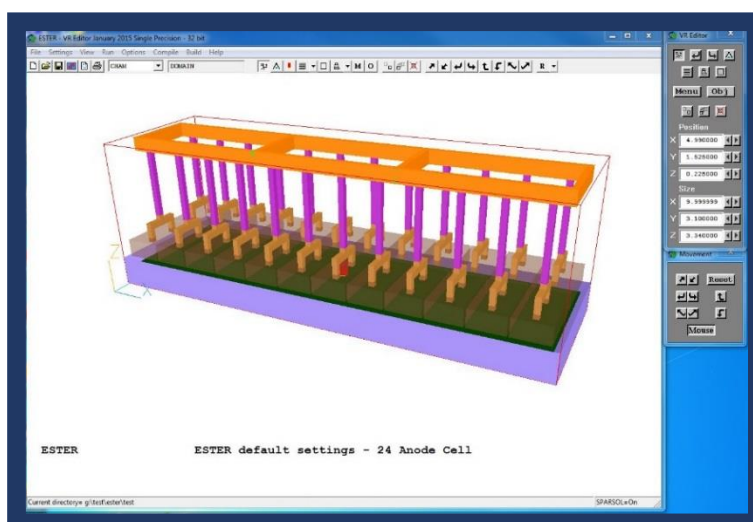
Corrections to PMV calculation to ensure agreement with ISO 7730.

ESTER

The [ESTER](#) user interface has been extensively upgraded to include:

- Cathode and collector bar configurations below metal pad
- Anode stubs, rods and bus bars above electrolyte layer
- Additional post-processing output, including
 - Interface height surface
 - Interface height map
 - Voltage and current monitoring files
 - Macros to create vectors from currents, magnetic fields and Lorentz Forces

The next image shows a typical ESTER setup:



CVD

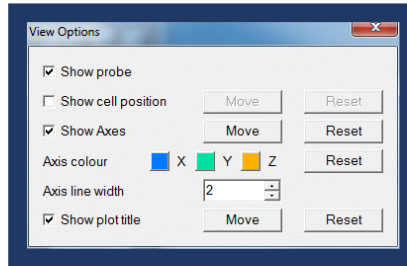
Allow blank and comment lines (starting !) in THERMDAT.


Allow blank lines in CHEMIDAT (which already allowed ! comments).

Increase the maximum number of reactions to 100

General, VRE & VRV

The colour and line width for the domain and object axes can be set (and saved to Viewer macro):

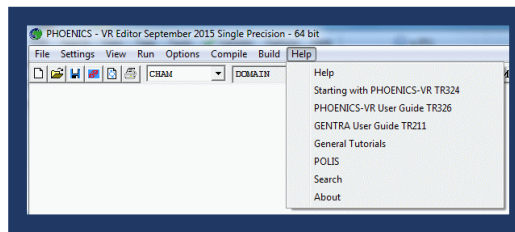


The toolbar and handset icons have been updated for better legibility when using Windows high contrast theme. The toolbar icons now look like this .

A new PHIDIFF utility for subtracting two phi(da) files. Useful for identifying differences between two solutions. It is run from 'Run / Utilities'.

A new PHISUM utility for summing contents of several phi(da) files. Useful for summing average velocity over several wind directions. It is run from 'Run / Utilities'.

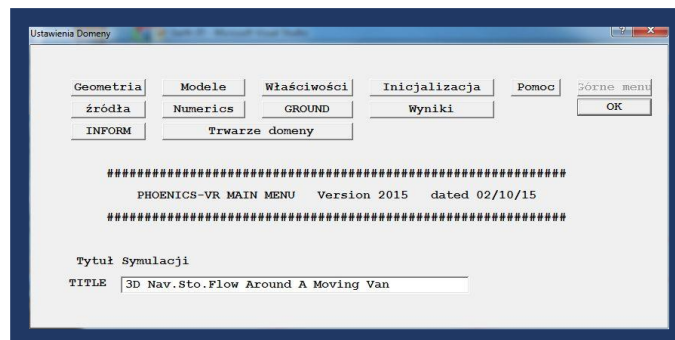
Tutorial and documentation files can be specified in phoesav.cfg. These then appear in the 'Help' menu:



Dialogs can be displayed in other languages if a language file is present. The image shows an example of an incomplete Chinese menu:



or in Polish:



Only licensed SPPs can start.