PHOENICS 2020

PHOENICS, the original commercial CFD Code, was brought to the market by Professor D Brian Spalding FRS, FREng in 1981. It has been regularly upgraded over the last 39 years with the current release, PHOENICS-2020, issued in July by the CHAM team based in Wimbledon Village.

PHOENICS has been used, continuously, worldwide for a range of applications relating to CFD which has, over time, become an essential tool for Engineers working in the general or built environment, green energy & industry not to mention those involved in research & academe.

PHOENICS models & simulates fluid flow, heat & mass transfer, combustion & chemical reaction. We work to keep it easy-to-use & updated.

This release contains improvements which the team considers beneficial; plus useful updates requested, or required, by PHOENICS Users.

CHAM's skilled Consultancy team can assist those with CFD needs who do not have the expertise, or desire, to carry out modelling in-house. Talk to us about your problems. We will work with you to reach optimum & economically viable solutions (consultancy@cham.co.uk).

We offer PHOENICS on heavily discounted terms to Academic Users. For full information contact sales@cham.co.uk.

Please note CHAM is unlocking PHOENICS for use at home during lockdowns. To avail yourself of this facility please contact phoenics@cham.co.uk

For general assistance & information email phoenics@cham.co.uk or phone +44 208947 7651.

What is New in PHOENICS 2020
VR-Editor

Improved Foliage Model:
Evaporation & Mass Transfer

CHAM has implemented a new system for analysing cooling and humidity effects from vegetation to assist Urban Planners, Architects & Civil Engineers to determine urban vegetation impact on pedestrian comfort.

The PHOENICS "Foliage" object could specify resistance of trees and bushes to prevailing wind. It now has the additional optional capability of specifying, within the defining attributes of the object, the EVT mass transfer rate of moisture from leaves to air, and accompanying (negative) heat source. This means that, for given species of tree, water supply and world location, architects and planners can simulate different arrangements of greenery within a site and determine the resulting impact on the urban environment.
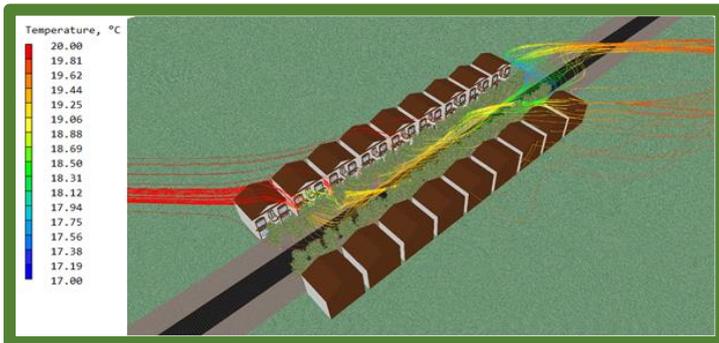
Streamline plot, coloured by temperature, showing the origin and destination of flow along the centre of the street at pedestrian level. Scale: Red – 20 °C; Blue – 17 °C
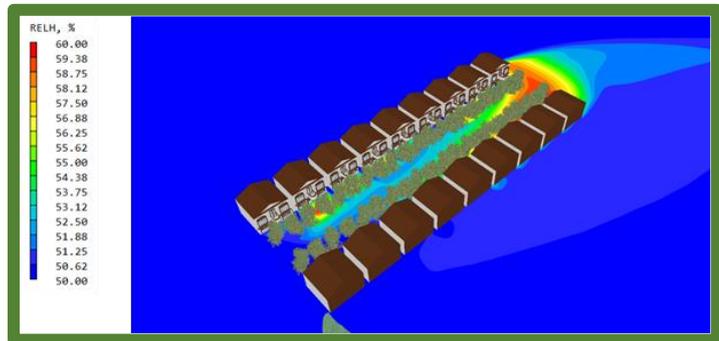
Contour plot of relative humidity at pedestrian level showing up to an increase of 10% humidity towards the leeward end of the street. Scale: Red – 60 %; Blue – 50 %
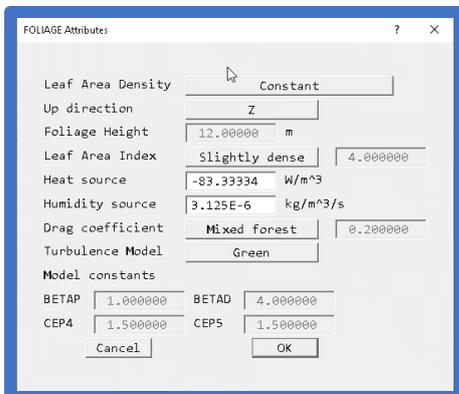
A pair of input boxes added to the Foliage object represent a group of trees, or other vegetation, to allow entry of the cooling rate in W/m^3 and water vapour release rate in $kg/m^3/s$. The 'Heat source' box should be set to the cooling power of the foliage (typically 250 W/m^2) multiplied by the Leaf Area Density. The 'Humidity source' box (only available in FLAIR) should be set to the moisture source in $kg/m^2/s$ (of the order 810 $g/m^2/d$), also multiplied by the Leaf Area Density.
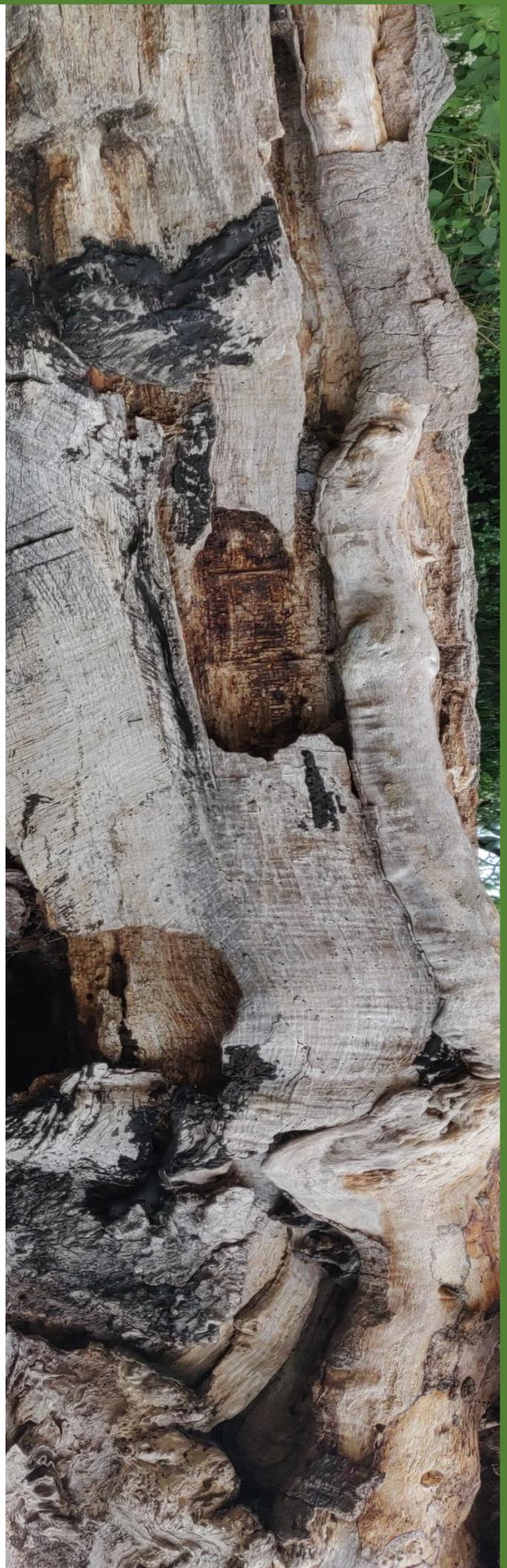
Display of Moving Domain

MOFOR, the moving-objects option in PHOENICS, has been able to impart motion to the entire domain for some time. For example, it was possible to model an oscillating tank.

The solutions were correct but it could be difficult to display them in the correct orientation. This is fixed.

Viewer will automatically display moving domains in their correct position. The images show a few time steps from a VOF simulation of an oscillating tank.
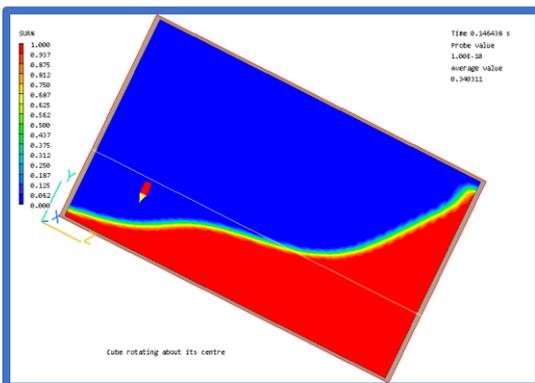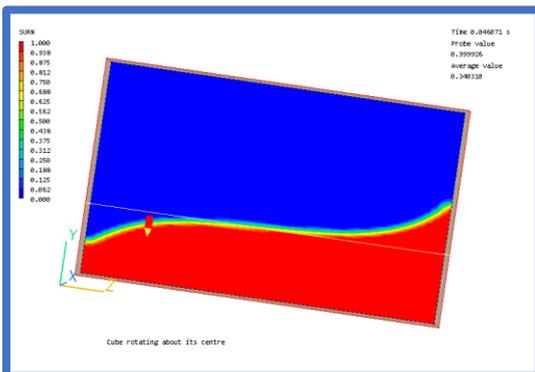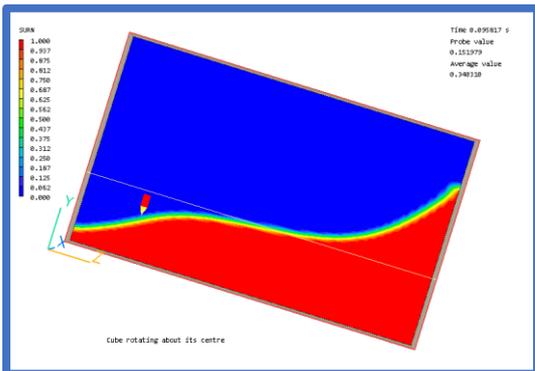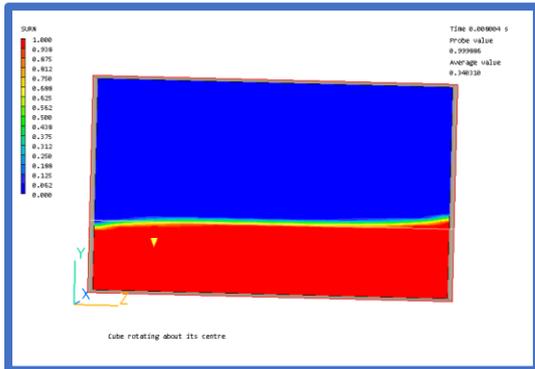
Clean Up List of Object Types

The list of object types presented when creating a new object has been re-ordered, and several object types deprecated, to simplify the user interface when creating new models. Object types are now grouped by:

- Creating blockages to the flow-volumetric (BLOCKAGE) or area (PLATE, THINPLT)
- Applying mass-flow and pressure boundaries (INLET, ANGLED-IN, FAN, OUTLET, ANGLED-OUT, WIND)
- Applying various sources (SUN, HEAT_SOURCE, FOLIAGE, USER_DEFINED, ROTOR)
- Miscellaneous.

Deprecated object types found in a Q1 are recognized and added to the list. Deprecated object types are: WIND_PROFILE, CELLTYPE, PCB, DRAG LIFT.
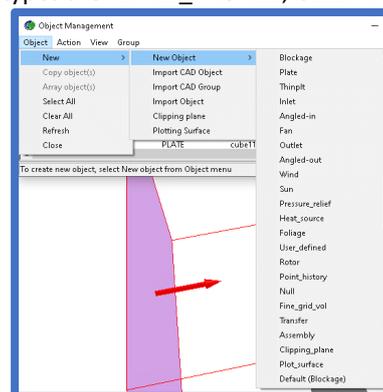
Change Default Object Geometry

Default geometry for many objects is changed from a form of 'cube' (eg cube14 for BLOCKAGE or cube3t for INLET) to 'box'. The impact is that all objects will, by default, be treated as faceted, so the subtraction of a domain-fluid blockage from a domain-filling solid blockage will always work first time.

Default colours and transparencies of object types are held internally, not taken from the object geometry. Each colour/transparency combination required a different geometry file hence the proliferation of cubes of different colour and opacity. The different cube geometry files have been moved to a deprecated folder so that existing Q1s will behave as before.
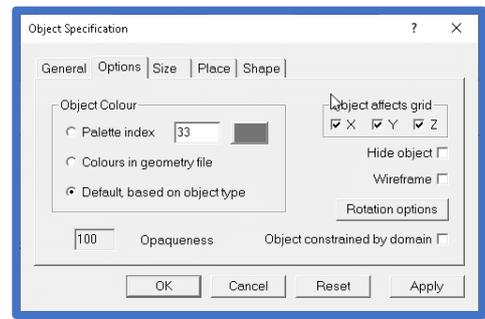
The new practice is reflected in the 'Object colour' dialog above. When a CAD file is imported, default is switched to 'Colours in geometry file', to ensure that any colour information held in the imported CAD file is acted upon.

Q1 PIL & InForm

New PIL/InForm Functions
RANDOM & MOD Added

The RANDOM(seed) function generates a random number based on a seed. The MOD(arg1,arg2) function returns the remainder of arg1 divided by arg2. Both functions are provided in PIL and InForm.

The RANDOM function is useful in generating boundary conditions for LES models. If the seed is zero, each call generates the next random number in the sequence, but the sequence of random numbers will be the same. The same non-zero seed value will always produce the same random number.

MOD function is useful in controlling how frequently expressions are evaluated.

Improve Viewer Startup Speed

Viewer animation creation can be slow. For each frame, an entire phida file must be read prior to a frame being drawn. Reading the minimum number of variables required for a current image, not an entire phida file, can speed this up.

Improved Ability to Interrupt Viewer Animation Creation

When Viewer is running an animation using large solution (phida) files, it becomes difficult, if not impossible to interrupt the animation. The animation sequence has been recoded to make it easier to interrupt an active animation.
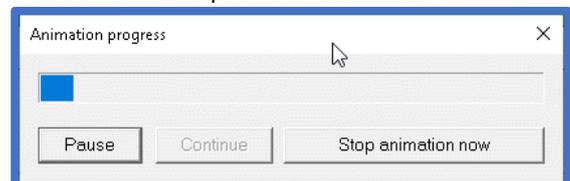
The above dialog appears once an animation starts, showing progress. Press 'Pause' to pause the animation until 'Continue' or 'Stop animation now' are clicked. 'Stop animation now' stops the animation as soon as possible.

Corrections to Viewer

When SPARSOL is active, and there are over 50 SOLVEd/STOREd scalars, cut-link information is written correctly by Earth to the solution file and read correctly by Viewer. Viewer will not crash, or display incorrect contours in cut cells.

All GENTRA produced tracks are drawn, even when Viewer displays a large number.

Solver – Earth

Automatic Calculation of Time Step

A 'Time step settings' option allows Earth to calculate the time step size itself. Dialog sets size of time step 1. The Flow field is scanned at time-step end for the largest Courant Number and the next step size is adjusted to maintain 'Target Courant Number' within user-set limits. The run stops when 'Last step number' is performed, or time exceeds 'Maximum total time'.

Current sweep and time step numbers, and time, are written to any dumped solution files enabling restart runs to continue smoothly and allowing Viewer to display the correct (calculated) time for each time step plotted. This benefits many transient, including VOF, cases where the process timescale changes during calculation.
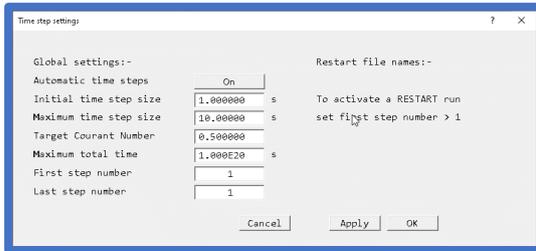
Outputting Forces & Moments per Object as well as Total

In addition to pre-existing files containing total-force evolution on all objects with time (or sweep) an additional file is output for each object participating in force summation.

Double Precision as the Default Solver

Experience shows double-precision use can result in much smaller residuals, with some cases reaching cut-off level. VR-Editor 'Options–Solver Precision' is defaulted to 'Double Precision'; the setting is held in PHOENICS.CFG. When restarting a pre-2020 case in 2020 it is advisable to delete local PHOENICS.CFG (& CHAM.INI) files to benefit from new default settings.
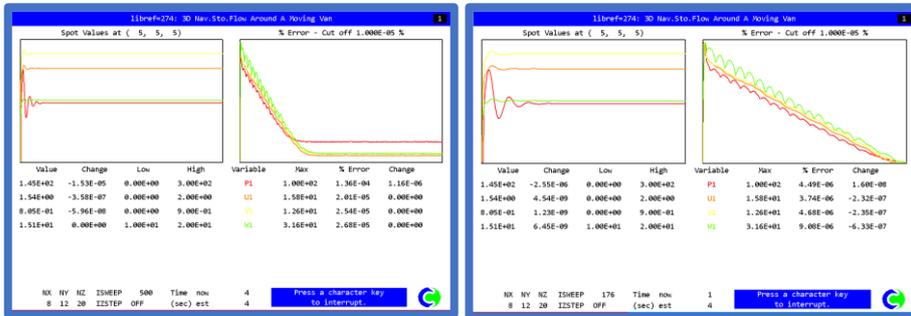
On the left, the single-precision solver reaches round-off error in the residuals but continues to solve for all sweeps as cut-off has not been reached. On the right, the double-precision solver reaches cut-off residual value and terminates. This occurs roughly after the same sweep number at which the single-precision solver reached round-off.

GXMONI – New combination of Monitor Graphs

An optional fourth monitor screen shows maximum absolute correction (left) and logarithmic residuals plot (right). It can be activated from VR-Editor 'Options–Solver monitor options' or from Q1 by setting ISG52=3. All monitor screens are dumped at run end; monitor display can be switched between any screen during a run.
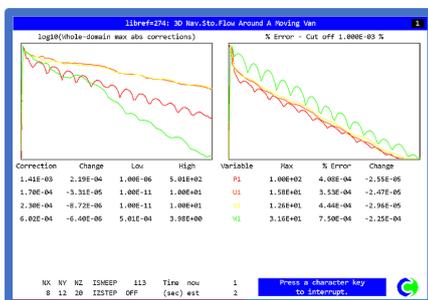
Flair / Flair-EFS

Allow use of Weibull Wind Coefficients for Averaged Wind Field

Flair / Flair-EFS have an option to calculate wind field and comfort index for a range of wind direction, and produce an averaged result.

Wind speed data were restricted to using wind speed and probability tables (WAsP tab format). This has been extended to allow wind date to be set using Weibull coefficients which can be read from a Generalised Wind Climate file (WAsP lib format).

Wind data format	WEIBULL
	Get Generalised Wind Climate File
Wind data file	gwa3_gwc_78pqdqcq.lib
Roughness class	0.030000 m
Height above ground level	10.00000 m

A link to the Global Wind Atlas website is provided.

From there, wind data for large parts of the world can be downloaded quickly in the form of a Generalised Wind Climate (GWC) file.

Lawson Criteria Settings

The number of Lawson Criteria bands has been extended to 7, though only the first 5 have default non-zero values. If more than 5 bands are required, it is up to the user to ensure that the probability and wind speed for each band is set correctly.

Dutch and Belgian Fire Standards

FLAIR's Fire Object and Smoke Solution panels have been extended to include Dutch NEN 6098 and Belgian NBN S 21-208-2/A1 Fire Standards.

The 'International' fire and smoke model, used in FLAIR for many years, can be used for most cases where production rates of heat and mass can be prescribed. Dutch and Belgian models are designed for simulating car fires in parking garages and hence are more prescriptive in the values used for heat of combustion and heat release rate.

Selecting the Dutch or Belgian model now automatically sets all parameters according to the chosen standard and displays them in the form used. This avoids the need to manipulate the International model into the form required by Dutch or Belgian Standards by setting parameters and heat release rates appropriately which requires some User knowledge of fire and smoke models.

The Dutch Standard specifies the heat release curves for 1, 2 or 3 cars on fire, with a time delay between the second and first, and third and second cars catching alight. The Belgian Standard is similar, but for one or two cars.

To calculate visibility through smoke International and Belgian standards use a 'Mass specific extinction coefficient', with a default value of 7600m^2/kg. The Dutch standard uses a 'Smoke Potential' with a default value of 400m^2/kg.

Heat and mass-release options for the FIRE object are restricted to those described in the standard.

The Dutch Standard specified heat-release curves for 1, 2 or 3 cars on fire, with a time delay between the second and first, and third and second cars catching alight. The Belgian Standard is similar, but for one or two cars.
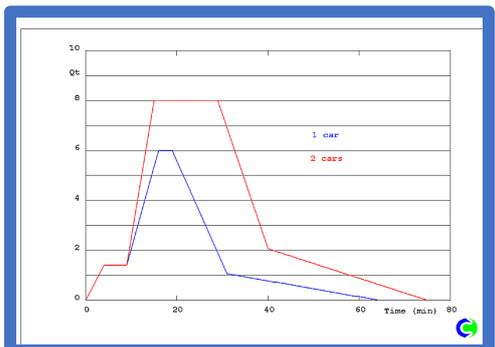
The mass release rate in both cases is then related to the heat release divided by the heat of combustion

General

Editor Corrections

Symmetric grid compression towards the domain edges in the first and last region has been turned off in the WIND object, leaving an expansion towards the domain edge.

An internal tolerance in the auto-mesh generation algorithm has been tightened to allow tiny regions with more than 15 cells.to be picked up.

An error writing the FACETDAT file in polar coordinates, so that some objects which lay over the polar axis were not detected correctly by the EARTH solver, is resolved.

Net area ratio For INLET objects is accounted for when deducing the velocity used to calculate inlet turbulence values to ensure consistency with velocity used for the momentum source.

Saving the Main Menu 'More' / 'Less' flags interfered with saving the 'Additional Interfaces' set on the 'Options' menu. If the mode was changed from 'Less' to 'More', the last active item in the 'Additional Interfaces' list was deleted. This is resolved.

Autoplot Corrections

The X-Y plotting program, AUTOPLOT, is modified to allow:
- #, * and ! symbols in table files to mark comment lines.
- longer column headers (up to 20 characters) in table files.

Earth Corrections

Restart in Parallel

Restarting the Earth solution on many (often more than 8) processors could fail. The cause has been traced and repaired.

VOF Surface Tension / Contact Angle

The contact angle is calculated correctly at the far end of the domain so asymmetrical solutions for the free surface do not result. Routines calculating normal to free surface are not called as they too often resulted in excessive runtimes

IPSA + PARSOL

When IPSA / PARSOL were both active Earth forced whole-field solutions of R1/R2. The facility is disabled to investigate convergence problems. (SPARSOL/old-PARSOL are unaffected).

Transfer Objects

Transfer objects can be represented by a facetted geometry file, but must remain planar and grid-aligned. An Internal indexing error is resolved.

New Solvers

In 2019, new solvers included several from the HYPRE suite. Point-by-Point (PBP) was the default pre-conditioner for the new solvers but was not supplied as a solver in itself. This meant cases using new solvers for some variables, but PBP for others, did not have a correct solution for variables solved PBP. VOF / SEM cases could be affected, where the SURN equation denoting volume fraction of heavy fluid is solved PBP. A PBP solver is now incorporated in the suite of new solvers.

Wall functions in Mass-flow Cells

Where a mass-flow condition coincides with a wall-function, eg an inlet or outlet on a solid blockage face, wall functions are skipped as there is no surface to create friction. This was omitted for ANGLED-IN and ANGLED-OUT objects for SPARSOL when setting values for KE or EP, and calculating YPLS, SKIN, STRS, STAN and HTCO. As a result, inlet values of KE and EP were overwritten by wall-function values, regardless of inlet / external turbulence values specified. This is resolved. PARSOL / old-PARSOL do not share the error.

CVD

If a run terminated before the last sweep, thermal conductivity and specific heat fields stored in the phida file as CNDT / SPHT did not fill with current values. The fields update for each print sweep and each time solution files are dumped during a run.

Excessive Warning Message

Frequency of a warning message advising users to change geometry for an object from cuboid to box has been reduced to once per affected object to prevent huge RESULT files from being created,

FLAIR Corrections

Drift Flux Model deposition patches generated automatically surrounding a cuboid object are now positioned correctly avoiding deposition potentially being wrongly located. When a deposition patch and mass-flow condition coincided (eg an inlet on a blockage face) deposition was calculated although, strictly speaking, there is no surface on which to deposit at such locations. In such cells STOREd values of deposition rate (DEPn) were calculated using undefined variables, potentially resulting in Infinite values. Nothing relating to deposition in cells affected by a mass source is now calculated.