CHAM Product Update

Pioneering CFD Software for Education & Industry

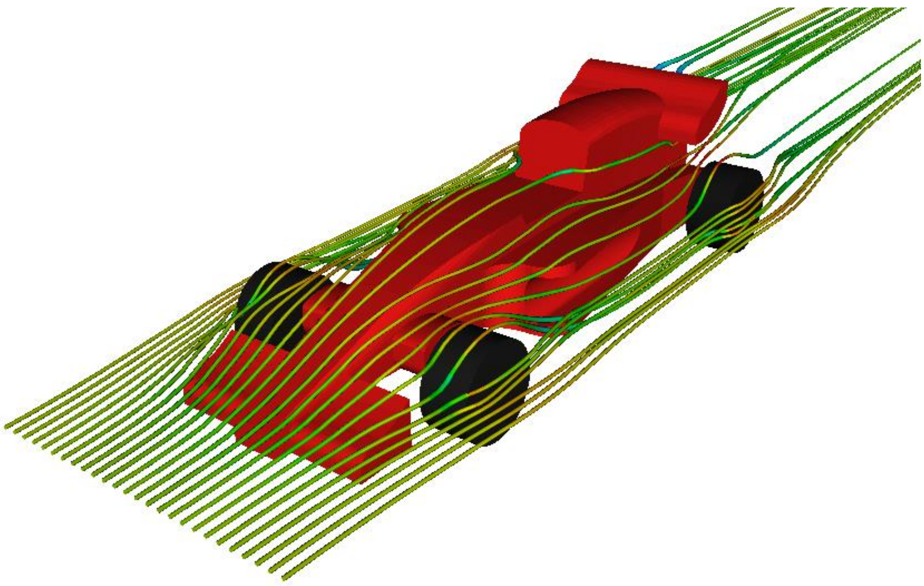
PHOENICS® to OpenFOAM® Interface [PH2OF]

PH2OF Documentation (JC 11/Jan/2016)

1. Overview

OpenFOAM is a free, open source CFD software package developed by OpenCFD Limited for the ESI Group and distributed by the OpenFOAM Foundation. – www.openfoam.com.

CHAM Limited has now added to its PHOENICS CFD package a special "PH2OF" (PHOENICS-to-OpenFOAM) 'translator' so that users are able to utilise the CFD solvers of both codes. This document describes the functionality of the PH2OF translator.



PHOENICS F1 VWT case replicated in OpenFOAM and visualised using ParaView

The translator can be run using two different approaches:

- From within the PHOENICS VR-Editor environment.
- Using the command line and the provided batch script in the working directory, without using the VR-Editor environment.

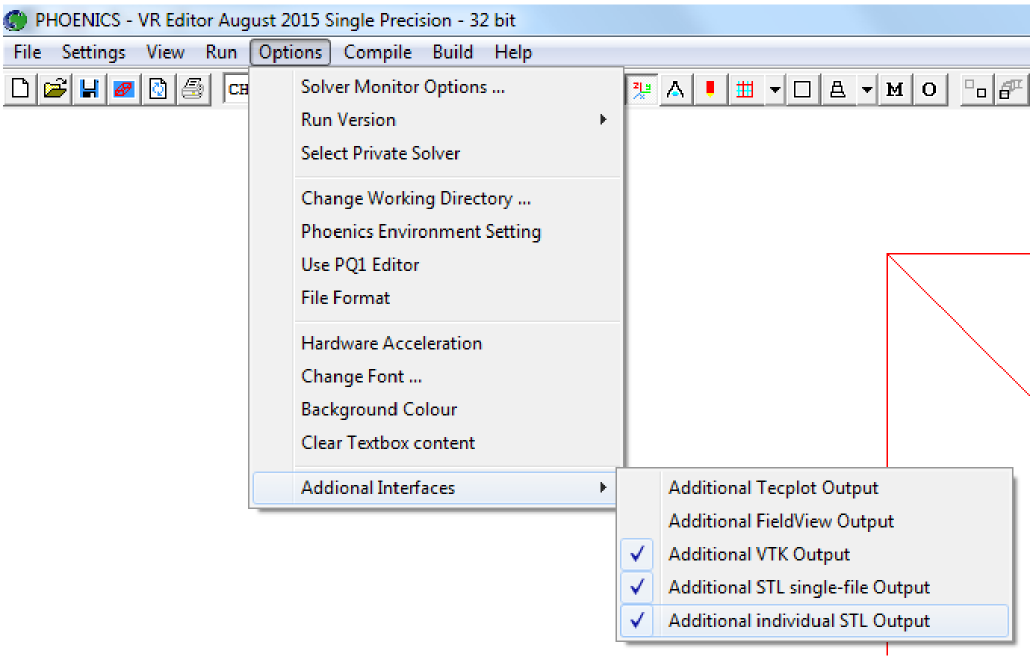
2. How to use PH2OF

2.1 Within VR-Editor

The simulation can be set up as a New Case in the usual manner within the VR-Editor. Alternatively, pre-existing library cases can be loaded from the PHOENICS libraries.

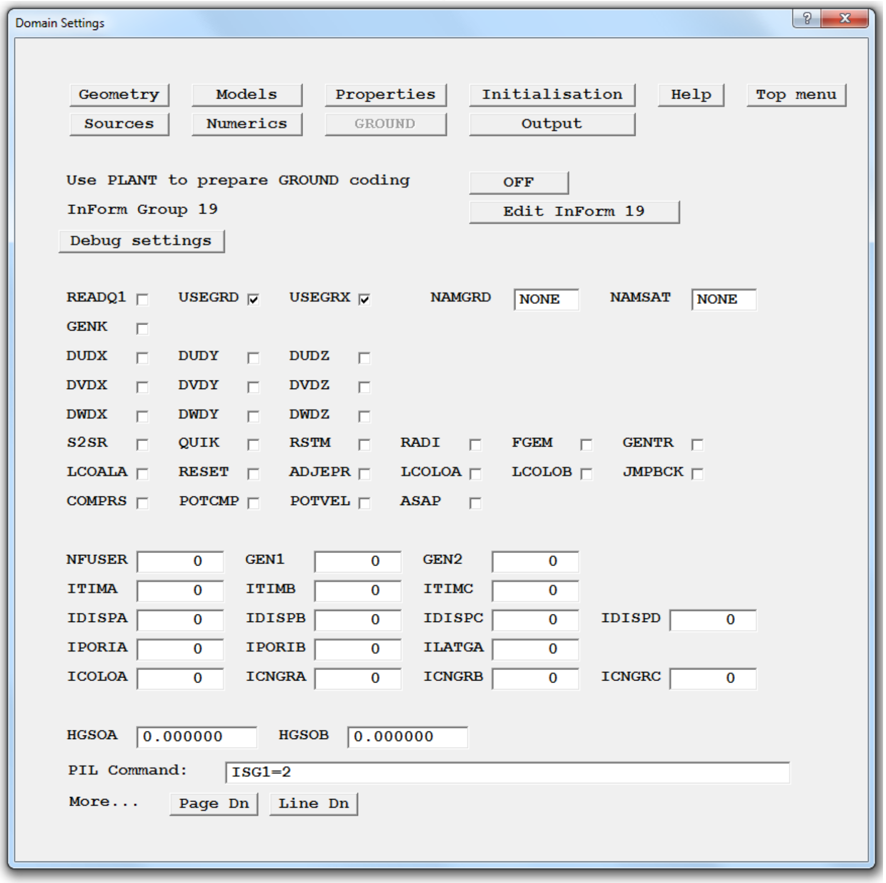
For cases that contain facetted objects (objects that have faces not aligned with the grid lines), it is important that 'Additional individual STL output' is activated in VR-Editor. Also, the probe monitor must be positioned within the fluid region of the grid.

Optionally, 'Additional STL single-file output' can be selected to output all domain objects in a single STL file, whilst 'Additional VTK output' will output the results of a PHOENICS-EARTH simulations in VTK format. This enables PHOENICS-EARTH results to be viewed in ParaView®, alongside OpenFOAM results. ParaView is an open-source, multi-platform data analysis and visualization application available from Kitware Inc – www.paraview.com.



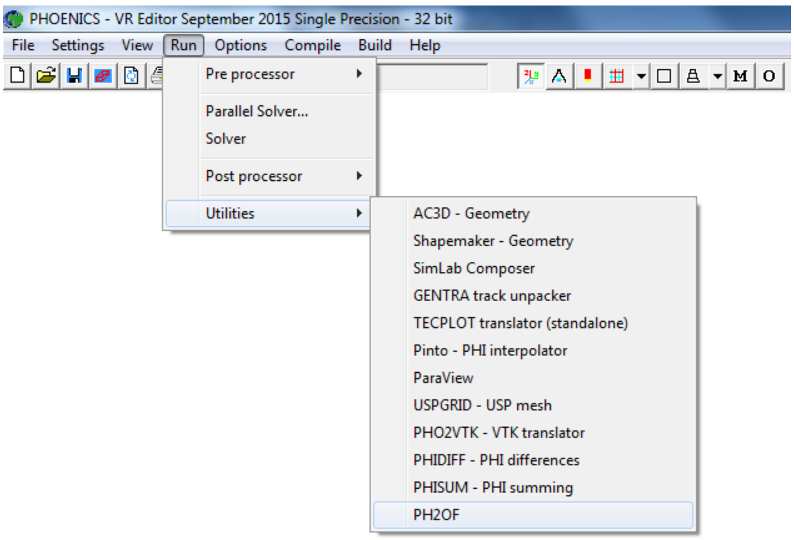
PHOENICS and OpenFOAM deal with facetted objects using different approaches. PHOENICS uses the PARSOL cut-cell technique, which automatically takes account of facetted objects on a structured grid.

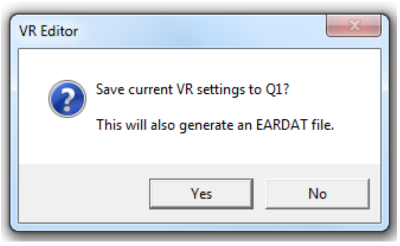
OpenFOAM adapts the VR-generated grid to fit the contours of a facetted object. There is an option to refine the grid near facetted object surfaces by setting the appropriate refinement level. This achieved by setting the variable, ISG1 in the 'PIL Command' box of the 'GROUND' tab within the PHOENICS VR-Editor (e.g. ISG1=2 for 2 levels of refinement). See below.



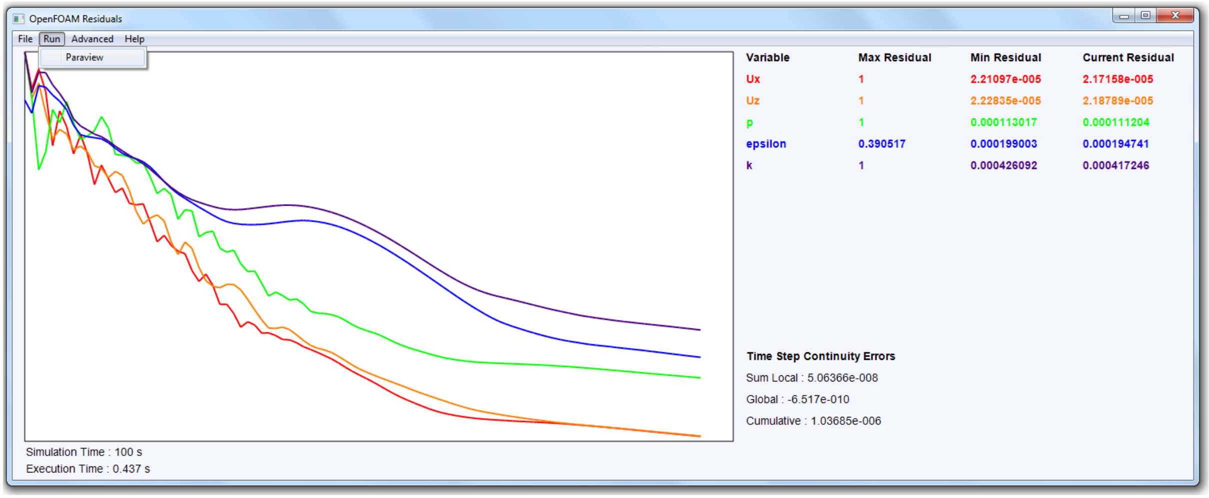
On completion of the case set-up, clicking on the menu item ‘Run/Utilities/PH2OF’ will execute the runOpenFOAM_Residual.bat script. The VR-Editor will ask for confirmation to save the EARDAT file. Click ‘yes’ to ensure that the EARDAT file exists or is up-to-date.

The batch script will run the PH2OF translator, producing the required OpenFOAM case files, and run the OpenFOAM solver whilst simultaneously plotting the residuals in the ‘OpenFOAM Residuals’ pop-up window.

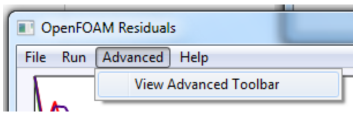




The OpenFOAM simulation can be aborted at any moment by right-clicking the mouse within the residual plot pop-up window. This will also write-out the final time-step data.

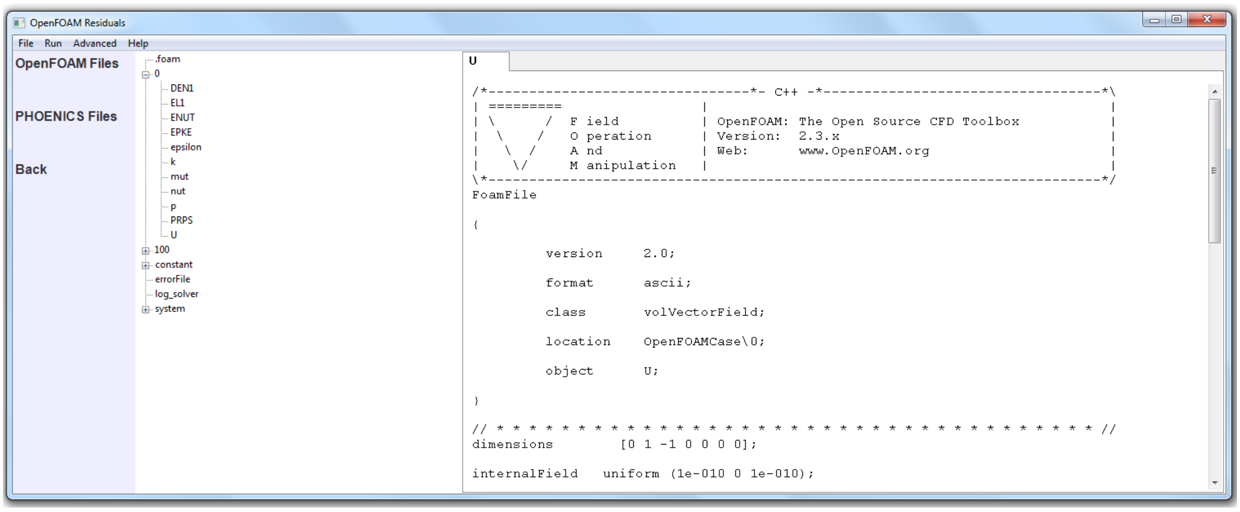


OpenFOAM files can be viewed by selecting the menu item ‘Advanced/View Advanced Toolbar’.



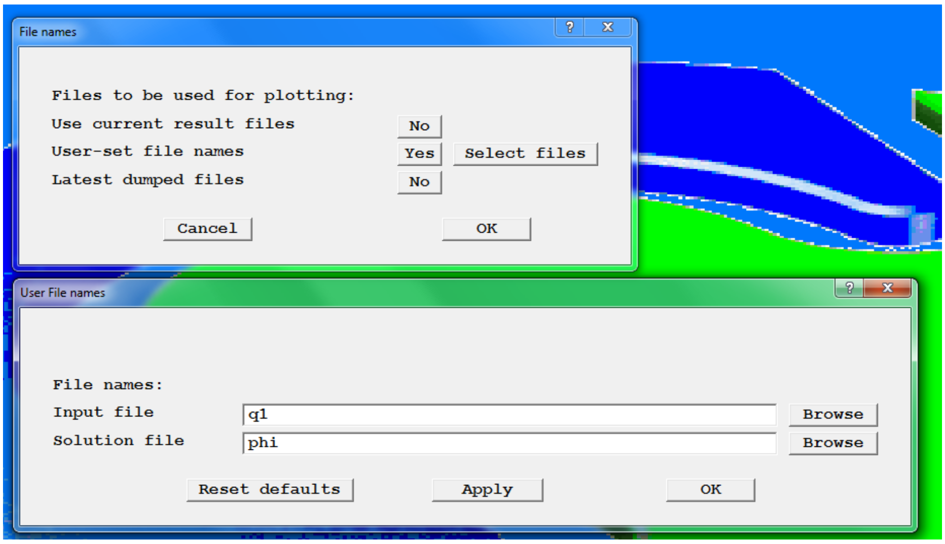
This causes a side-toolbar to open, with the option to also view the PHOENICS files. Double-clicking on a file item within the directory tree will enable it to be viewed within the window.

An important file is ‘errorFile’ which will display any errors that have occurred with the PH2OF translation process. The various log files that are produced by OpenFOAM, such as ‘log_solver’, contain information relating to the OpenFOAM simulation and are helpful in diagnosing any run-time errors.



2.2 Data visualisation

The results can be viewed within ParaView by clicking on the menu ‘Run/ParaView’ within the residual plot window. Alternatively, for simulations which contain non-facetted objects, a PHOENICS phi file will have been written to the working directory. This file can be viewed within the PHOENICS VR-Viewer by selecting ‘User-set file names’ and ensuring ‘Solution file’ is set to ‘phi’.



Note: At present, for simulations which contain facetted objects, the results can only be viewed in ParaView. This can also be achieved by selecting the VR-Editor menu option ‘Run/Utilities/ParaView,’ and within the ParaView environment opening the file OpenFOAMCase/.foam to view the current data.

2.3 Using the command line

PH2OF can be run without the need to enter the VR-Editor. In a working directory containing a valid EARDAT file, running the script ‘runOpenFOAM_Residual.bat’ will cause PH2OF to generate the required OpenFOAM input files and then run the simulation whilst simultaneously plotting the residuals. If an OpenFOAMCase directory already exists in the current directory it will be appended with the current date/time and another OpenFOAMCase directory created for the current simulation. Running PH2OF in this way, it is important to ensure that any facetted objects have been saved in STL format within the working directory and that the probe location has been positioned within the fluid region of the grid (by setting i,j,k values of the probe (IXMON, IYMON, IZMON)).

3. Files created by PH2OF

PH2OF will create a new directory called ‘OpenFOAMCase’ where all OpenFOAM case files are stored. In addition, batch scripts are created within the working directory to enable automatic running of simulations and post-processing of the results. Any errors that occur during the execution of the translator will be written to ‘OpenFOAMCase/errorFile’ as well as outputted to the console window during execution. Typical errors may be incompatible thermophysical models or invalid turbulence model selection.

4. PH2OF Functionality

This section lists the current functionality of the PH2OF translator.

Geometry

- Cartesian co-ordinate systems.
- Time dependence – Steady and transient.

Objects

- Blockage
- Inlet
- Fan – Rectangular fan only
- Outlet
- Plate
- Porous plates – Superficial velocity only, with pressure drop formulation (Velocity-squared, Power of velocity, Linear in velocity).
- Thin-plate
- Pressure-relief

Models

- One-phase only
- Scalar equation method for free-surface flows.
- Thermal modelling including conjugate heat transfer.
- Turbulence Models – Laminar, constant-effective, KEMODL, KOMODL.
- Transport of C1, C2 etc.
- No radiation at present.
- No combustion/chemical reaction at present.

Properties

- The majority of material properties can be translated from PHOENICS into OpenFOAM. However, certain configurations are not compatible. An error is outputted for incompatible OpenFOAM material properties and available options are suggested.
  (e.g. In OpenFOAM, if the density is defined by the ideal-gas law then the viscosity <u>must</u> be defined by Sutherland's formula.)

Sources

- Gravity – Constant, Boussinesq.

Numerics

- Linear relaxation only
- Upwind differencing scheme. Others, such as, QUICK, VANL1, LUS, UMIST, CDS can easily be included.

Helpful hints

- OpenFOAM often requires relaxation on pressure (P1) for convergence.
- Running PH2OF again in the same working directory will cause the old ‘OpenFOAMCase’ folder to be appended with the date and time (e.g. ‘OpenFOAMCase_08Jan16-142755’), and another current ‘OpenFOAMCase’ directory to be created.
- To diagnose errors first check the ‘OpenFOAMCase/errorFile’ file. If no errors are reported, check the OpenFOAM log files, such as ‘log_solver’ or ‘log_snappyHexMesh’.

5. Setting up OpenFOAM and PH2OF

1. Unzip ‘OpenFOAM.zip’ to the C:\ drive
2. Run MSMPISetup.exe
3. Install ParaView using ParaView-4.1.0-Windows-64bit.exe
4. Set the OpenFOAM and PH2OF environment variables by double-clicking on OpenFOAM-2.3.x-reg.
5. Add PH2OF menu option to VR-Editor by editing C:\phoenics\d_allpro\pheosav.cfg

   Add the following line below the last entry of '* List of Executable Files'

   UTILITY PROG = C:\OpenFOAM\PHTOOF\runOpenFOAM_Residual.bat 'PH2OF'

6. Acquiring the PHOENICS-to-OpenFOAM translator

PH2OF will become available to existing licensees of PHOENICS-2015 and as a download via CHAM’s FTP site.

CHAM Ltd, Bakery House, 40 High Street, Wimbledon Village, London SW19 5AU, UK
Tel: +44 (0)20 8947 7651 Fax: +44 (0)20 8879 3497 Email: phoenics@cham.co.uk
Web: www.cham.co.uk