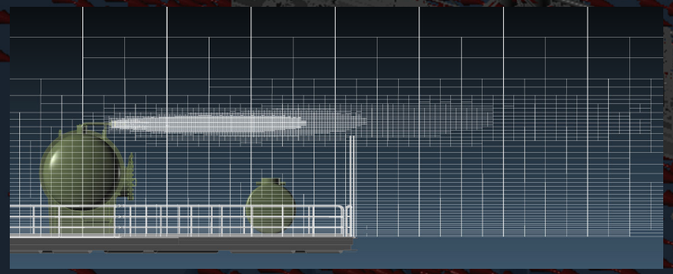
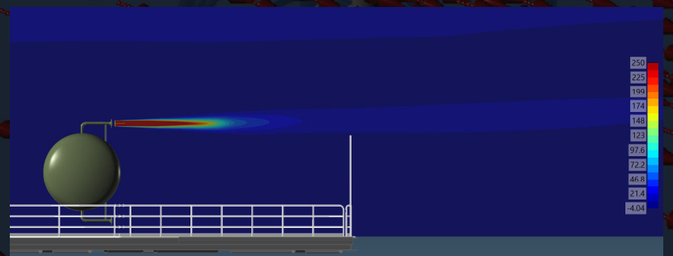
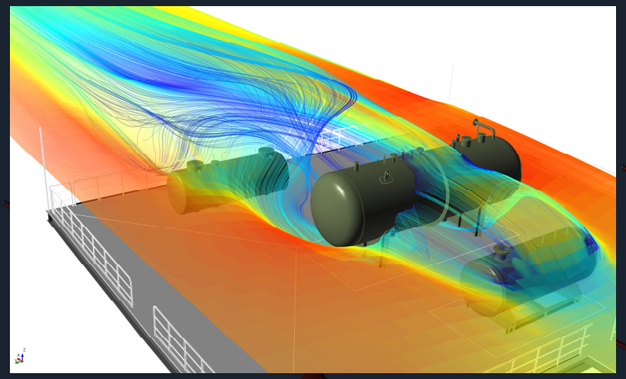
in:Flux

INTELLIGENT CFD SOFTWARE

in:Flux is a CFD software product that analyses fluid dispersion within complex geometries in a fully three-dimensional environment. It is aimed at engineers who have previously reviewed CFD results, but have not carried out the analysis themselves. **in:Flux** seamlessly adds a CFD capability to organizations that have previously out-sourced prior dispersion modeling projects refocusing the expertise requirements into engineering design instead of CFD.







For more information on **in:Flux** email us at info@insightnumerics.com

Featured in:Flux capabilities:

- Automated adaptive meshing and refinement
- Automatically defined domain sizing and boundary conditions
- Merge and transfer data with Detect3D (.d3d) project files to facilitate gas mapping studies
- Full CAD integration – no need to represent CAD with primitive shapes
- Calculate focused CFD results in minutes rather than days

in:Flux Benefits

Many of the tasks that normally require outsourced expertise are handled automatically by **in:Flux**. For example, meshing is entirely automatic – the mesh will refine around geometry and regions of high gradients. Numerical discretization schemes and physical models are all automatically set, as are boundary conditions and convergence criteria. The user is simply required to input the required wind speeds and directions, and point sources of gas emissions and wait for the solution to complete. This level of automation provides assurance of high quality CFD results.

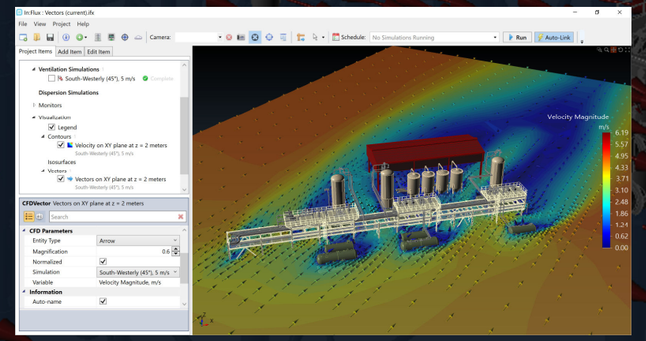
Once the simulation has finished, the user can add streamlines, contour plots and 3D isosurfaces to analyze the data. All the post-processing can be output to images files for reporting.

Technical Description

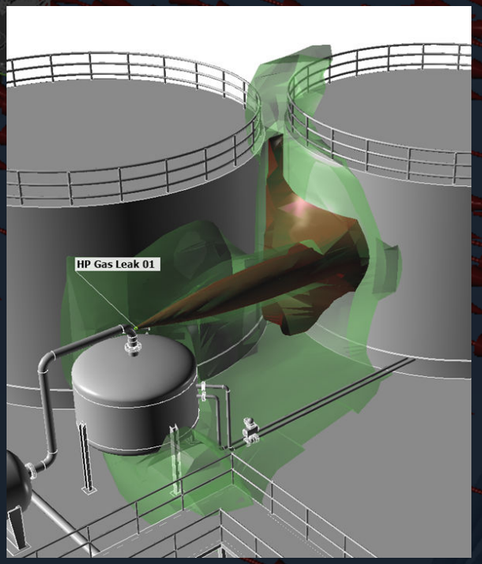
in:Flux solves the Reynolds-Averaged Navier-Stokes equations (RANS) on a mesh using a Finite Difference discretization. The mesh automatically refines around geometry and regions of high gradient. Geometry is accounted for using an Immersed Boundary Method (IBM).

The momentum and continuity equations are solved iteratively using the PISO algorithm until convergence is attained. Transport equations for turbulence, heat and concentration are also solved. Mesh refinement is part of this solution process.

Turbulence is modeled using the standard k-epsilon model with buoyancy production and dissipation. Within the momentum equations, buoyancy is added using the Boussinesq model.



in:Flux is developed and maintained by Insight Numerics. For further information, contact one of our offices. Details are provided on the back page.



Insight Numerics, LLC
Boston, USA
+1 (617) 657-4708
www.insightnumerics.com

insightnumerics