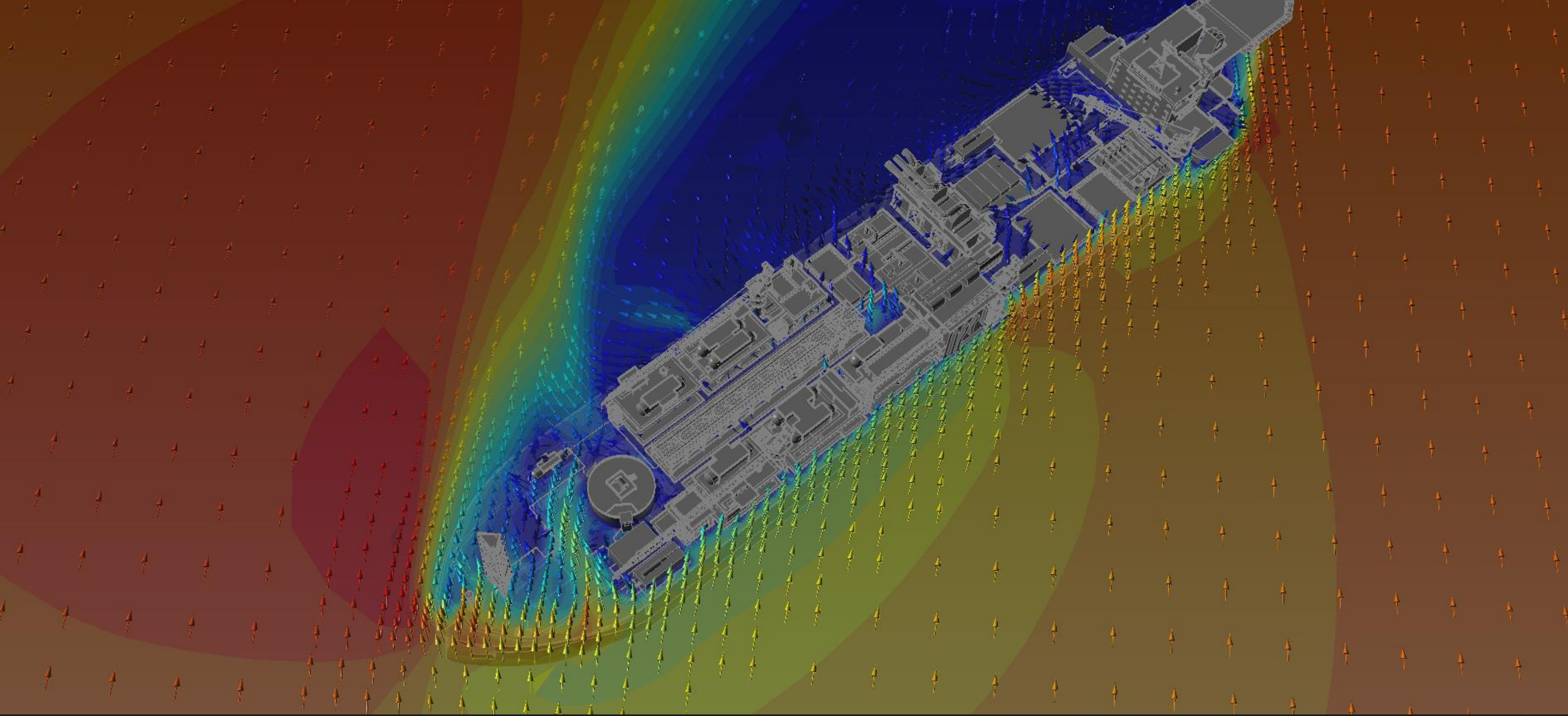


in:Flux - Intelligent CFD Software

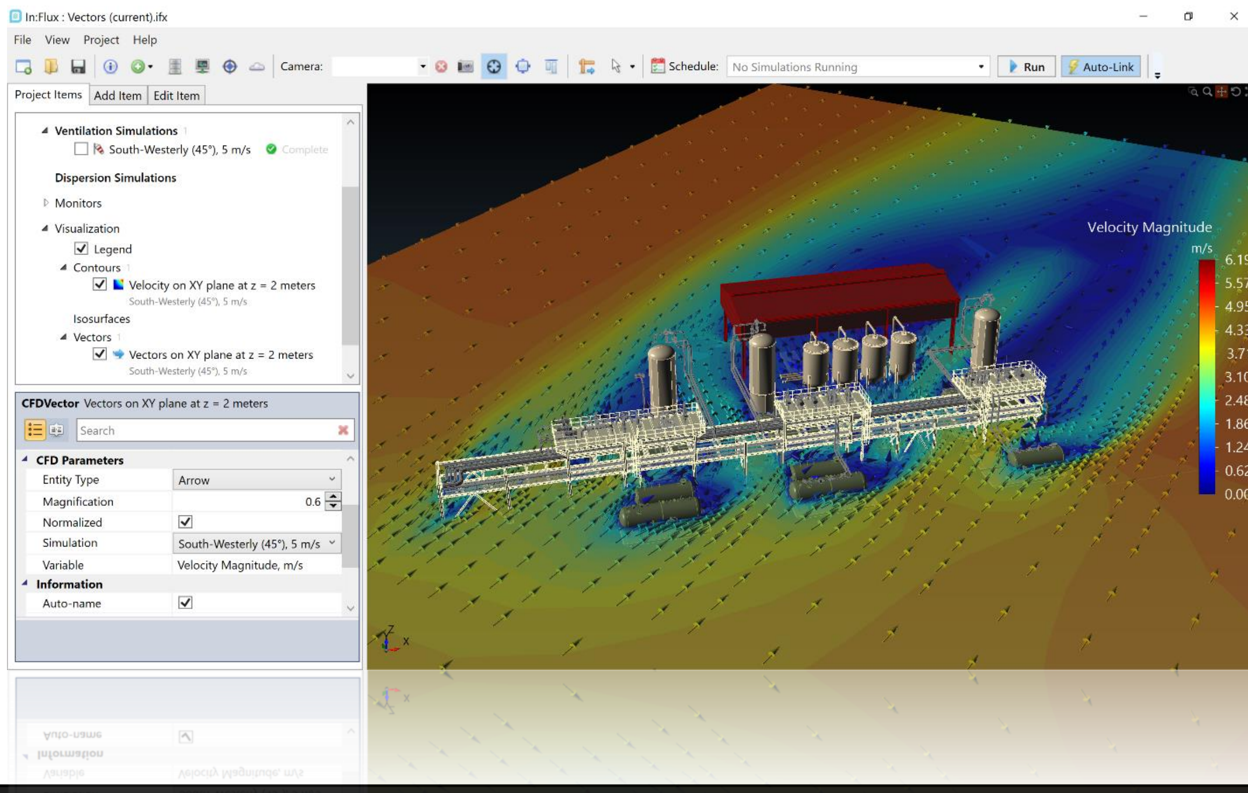
Developed by Insight Numerics

info@insightnumerics.com
www.insightnumerics.com

insightnumerics

Introduction to in:Flux

in:Flux is an automated and intuitive CFD software product to be used for dispersion and ventilation studies. It is developed entirely by Insight Numerics.

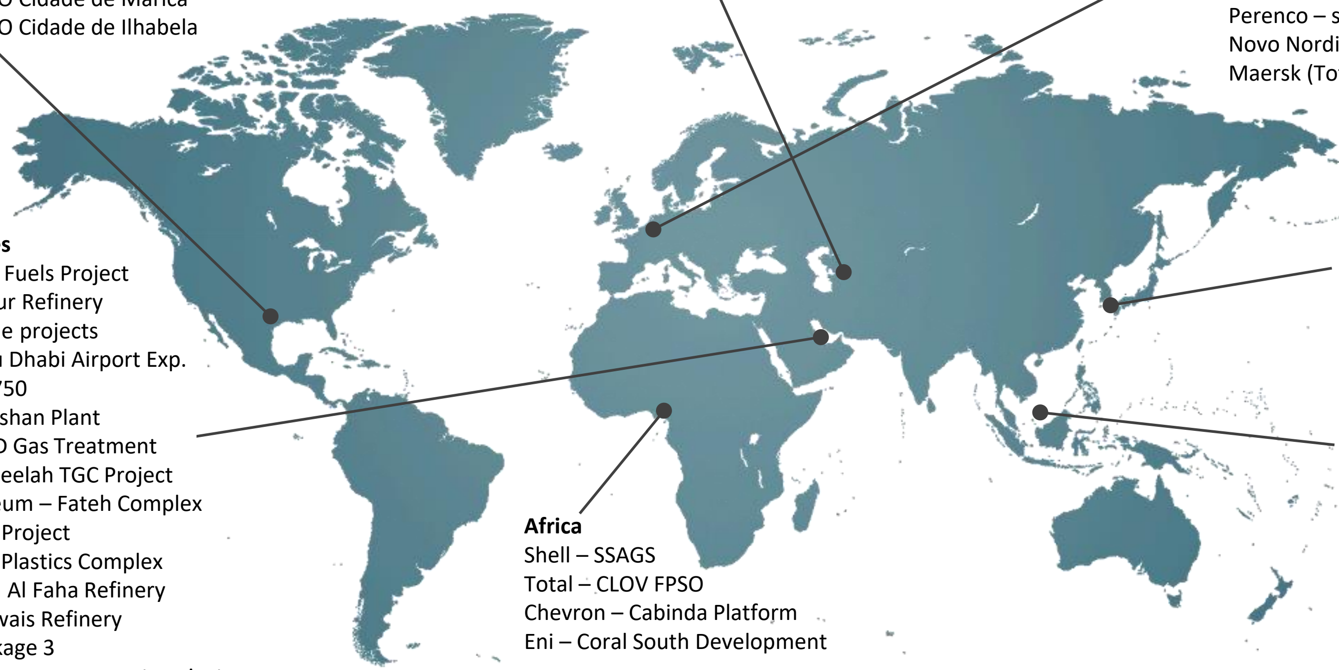


Who Uses Insight Products?

- Chevron
- bp
- FLUOR®
- Petrofac
- wood.
- JGC
- SAIPEM
- HYUNDAI HEAVY INDUSTRIES CO.,LTD.
- DNV·GL
- TechnipFMC
- KBR
- AkerSolutions™
- PERENCO
- SAMSUNG
- SAMSUNG ENGINEERING
- CHIYODA CORPORATION
- CB&I
- McDERMOTT
- ATKINS
- ptt ES
- Dome
- NOOC
- ABS Group
- DEKRA
- PRISM (Process Risk Integrity and Safety Management)
- CORE
- BMT
- SAFETEC
- arc
- BLACK & VEATCH
- NAPESCO نابيسكو
- Applus⊕ VELOSI
- Multiconsult
- Chola MS RISK SERVICES
- ae Solutions™
- ISPI INTERNATIONAL
- SYNERGY specialist engineering
- DET-TRONICS® A UTC Fire & Security Company
- Liquefied Natural Gas Egyptian LNG
- Techniche HEALTH SAFETY ENVIRONMENT
- Advisian WorleyParsons Group

Where is Insight Numerics used?

- Our software has been used on projects worldwide for BP, Shell, Chevron, Petronas, ADNOC, Petrobras, Maersk and many more.



The Americas
TechInt – Pluspetrol Plant
ALPEK – FGRU Indelpro
SOFEC – several projects
Petrobras/MODEC – FPSO Cidade de Caraguatatuba
Petrobras/MODEC – FPSO Carioca MV30
Petrobras – FPSO Cidade de Marica
Petrobras – FPSO Cidade de Ilhabela

Asia
BP – South Caucasus Pipeline Expansion
Chevron/TCO – Kazaskhstan Project
GAZPROM – Badra-Zubaida Pipeline

Europe
Equinor (Statoil) – Johan Sverdrup Platform
Maersk – Culzean Platform & FSO
Ineos – Grangemouth
Apache – multiple projects
Technip/JGC – Coral South Development Project
BP – West Sole Charlie Platform
Perenco – several projects
Novo Nordisk – API Manufacturing
Maersk (Total) – Tyra Platform

East Asia
COSCO Shipyard – FPSO
HHI – multiple projects
Seoul National University

GCC Countries
KNPC – Clean Fuels Project
KNPC – Al-Zour Refinery
KOC – multiple projects
ADNOC – Abu Dhabi Airport Exp.
ADNOC – UZ750
GASCO – Habshan Plant
GASCO – ICAD Gas Treatment
GASCO – Taweelah TGC Project
Dubai Petroleum – Fateh Complex
BP – Khazzan Project
ORPIC – Liwa Plastics Complex
ORPIC – Mina Al Faha Refinery
Takreer – Ruwais Refinery
DUQM – Package 3
EgyptianLNG – reassessment project

Africa
Shell – SSAGS
Total – CLOV FPSO
Chevron – Cabinda Platform
Eni – Coral South Development

SE Asia
PETRONAS - FLNG 2
ThaiOil – Clean Fuels Project
Vestigo - MaMPU FPSO
Pertamina – PPP
JX Nippon – Helang Project
HESS – Anggerik WHP Platform
Linde/Petronas – Marlin BOG LNG

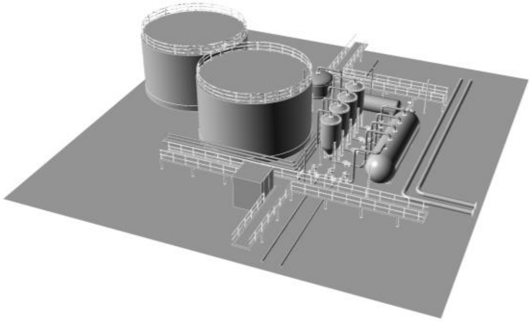
Why use in:Flux?

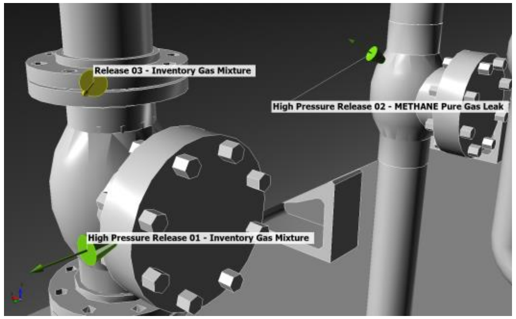
- Virtually all setup is automated, including domain sizing, meshing, boundary conditions, numerical setup and post-processing.
- Over 100 simulations can be set up in less than 5 minutes.
- in:Flux is very fast compared to other CFD software – ventilation simulations take 2-10 minutes, while dispersion simulations are typically 10-30 minutes.
- Once simulations have started, they require no additional monitoring.
- On a single analysis machine, more than 100 CFD simulations can be run in a single day.
- All simulations are saved in a single compact file, which makes data management easy. No CFD expertise is required – the automatic setup ensures very high quality CFD simulations.

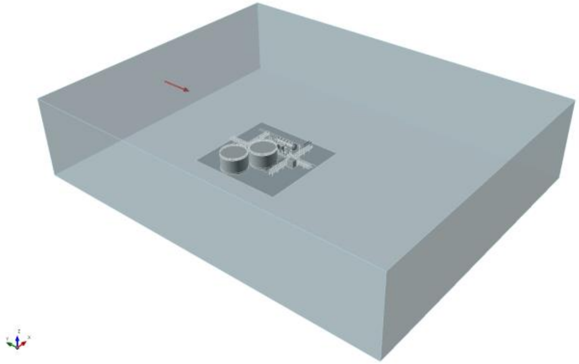
in:Flux Capability Overview

in:Flux Inputs

- The required user inputs are:
    1. 3D CAD file
    2. Knowledge of process conditions (pressure, temperature, inventory) and required leak locations.
    3. Knowledge of the site atmospheric conditions, such as a wind direction and wind speed
- All of the above user inputs are typically required for CFD projects, **even if the project is outsourced.**

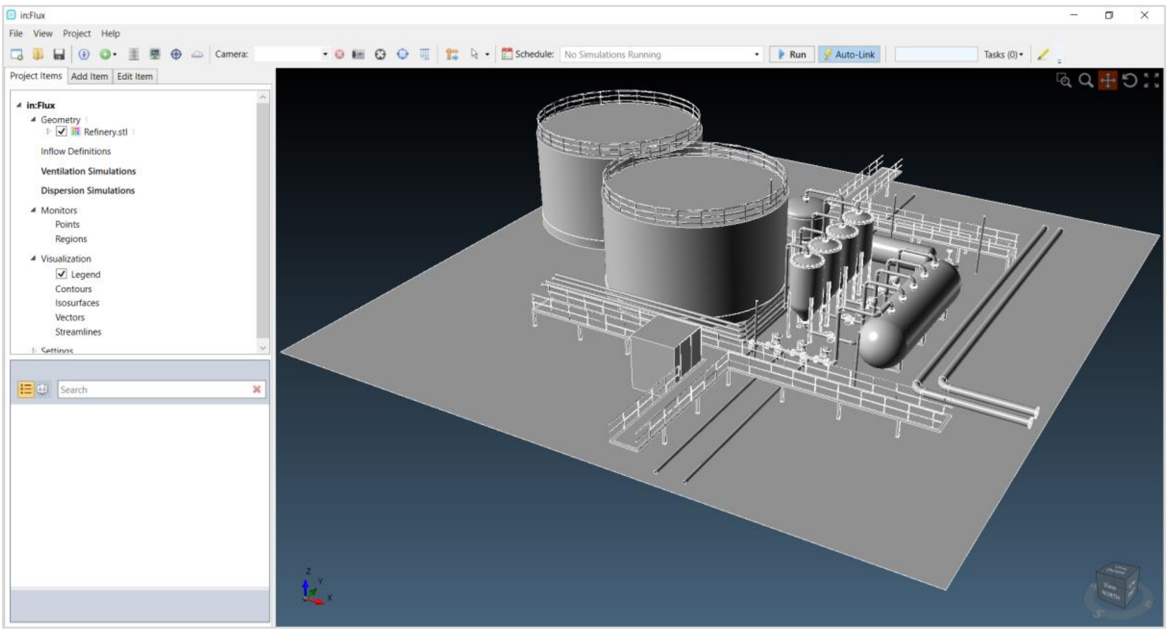


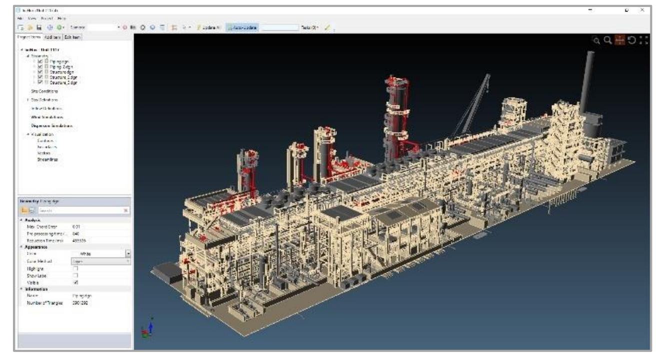


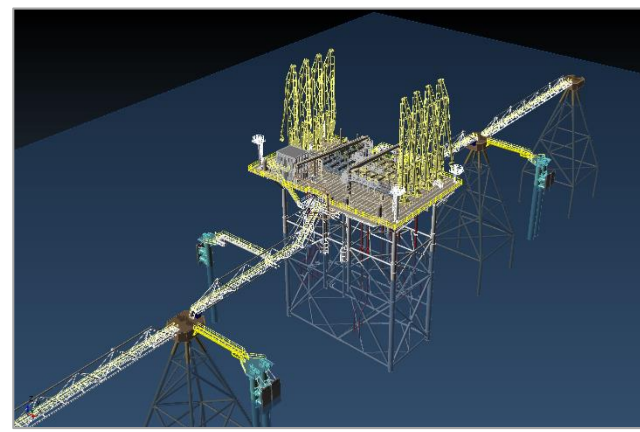


CAD Import

Via the DGN file import, in:Flux can load CAD from **PDMS** and **SmartPlant3D**. **Navisworks** (NWD) files can be imported via DWF files and AutoCAD DWG, DXF and other standard CAD formats (STEP, IGES, OBJ and STL) are also accepted.



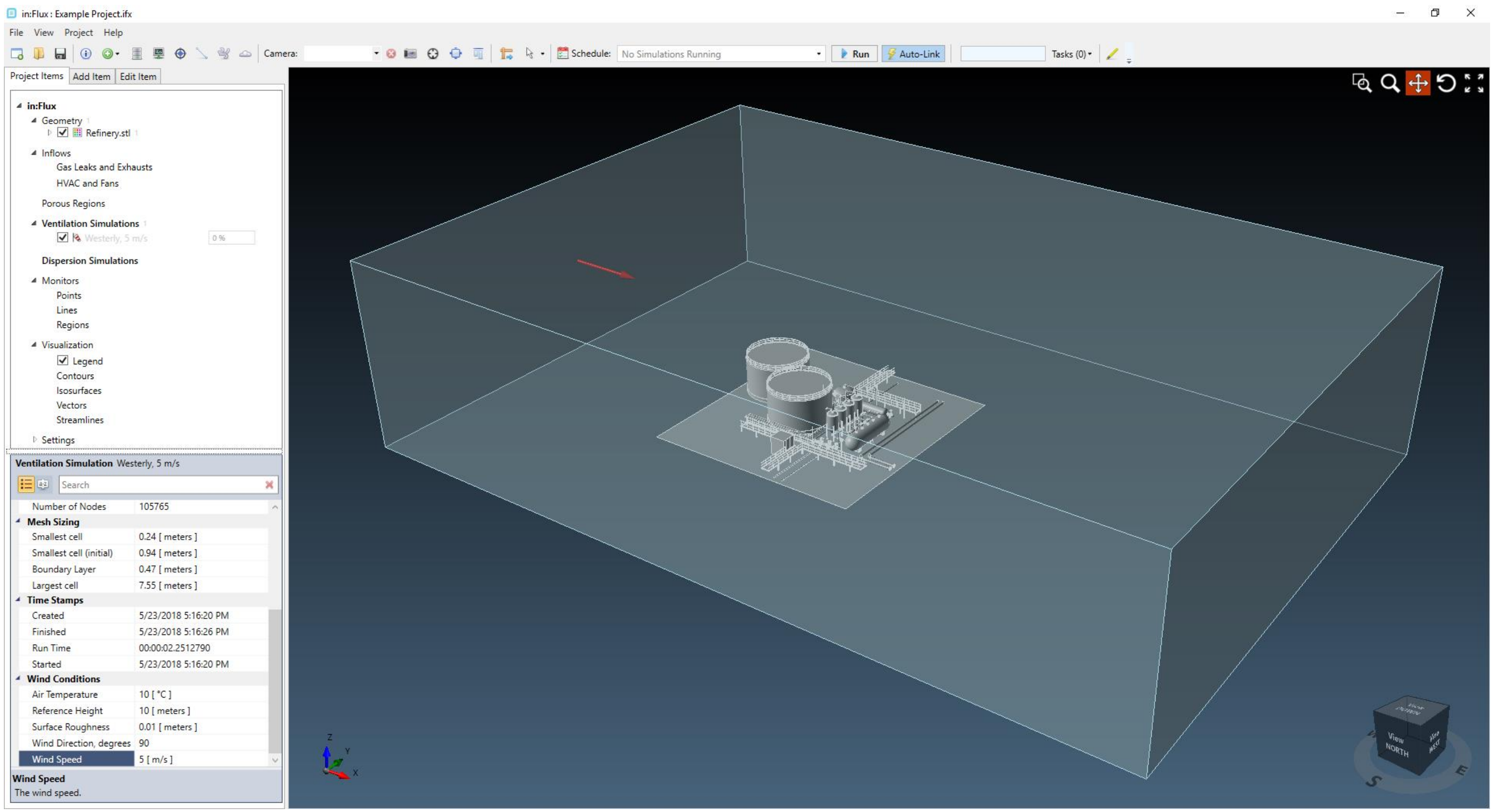




Above Screenshot of in:Flux showing a simple CAD file of an example facility.
Right Other examples of CAD models loaded into in:Flux. There is no limitation to size or complexity of the CAD model which can be loaded.

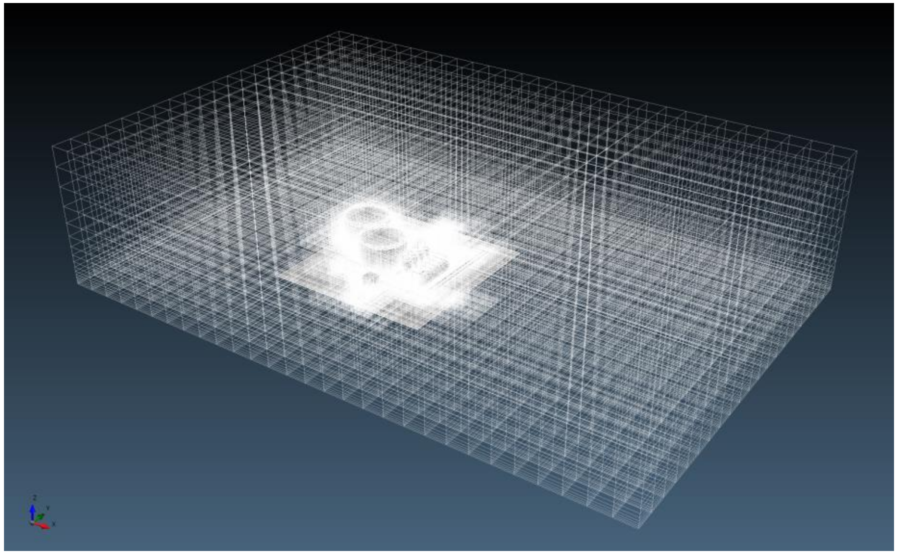
Automatic Domain Sizing

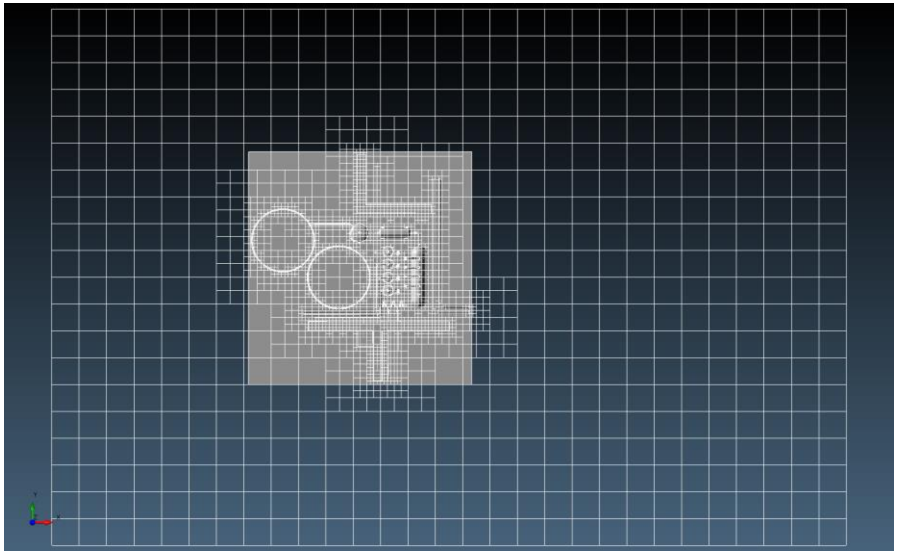
Once CAD has been loaded into the software, ventilation simulations can be defined simply by choosing a wind direction and wind speed.



Automatic Meshing

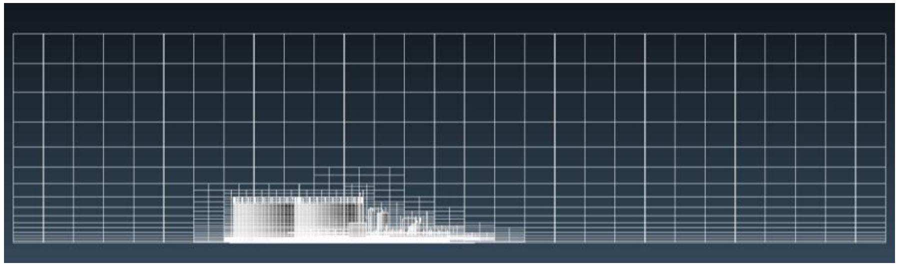
The mesh initially refines around the geometry, and includes boundary-layer grading from the ground to capture the high gradients.





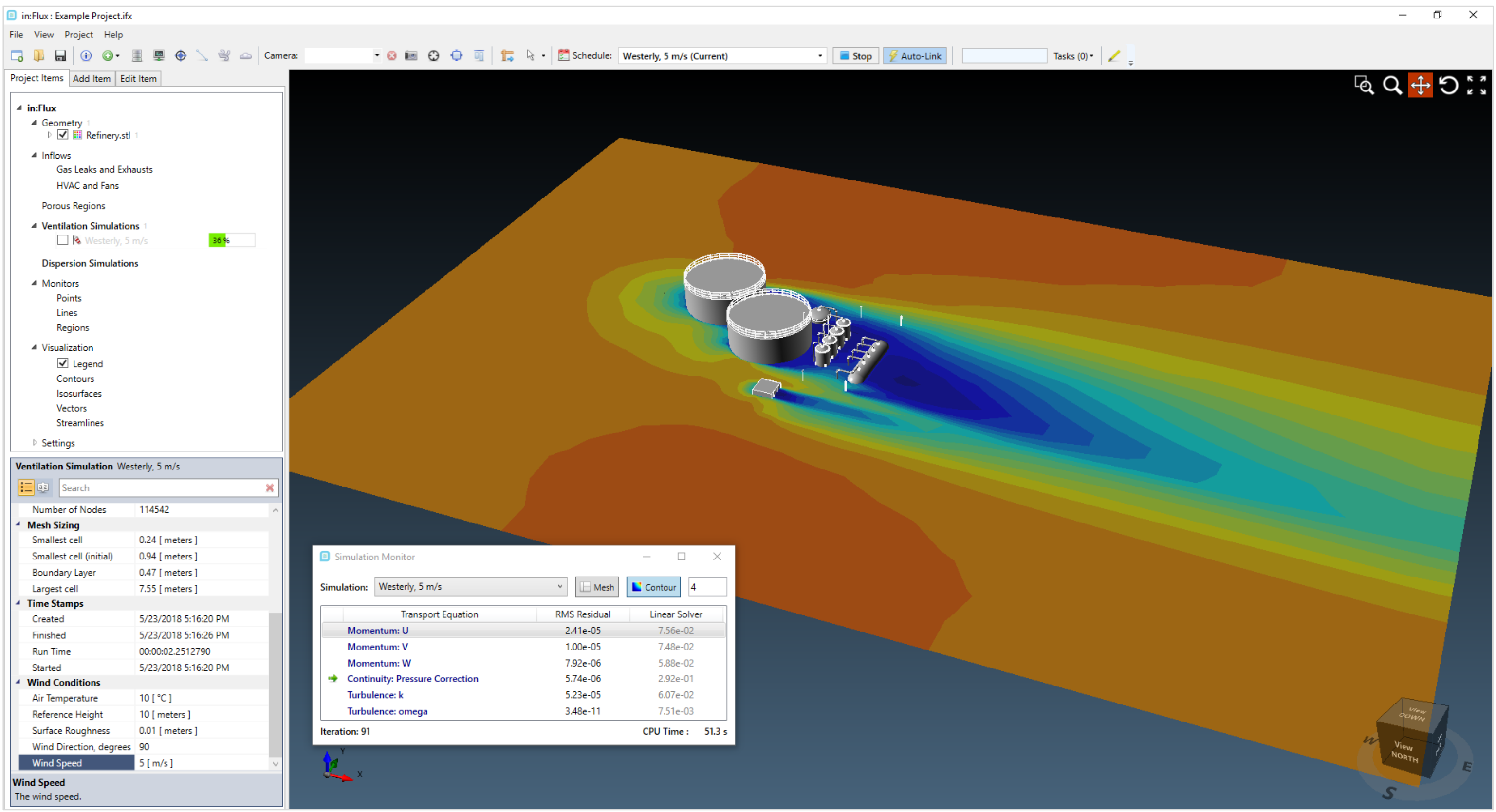
Above and Upper Right
Top and isometric views of the mesh around the CAD model.

Right
Side view of the auto-generated mesh around a CAD model. Showing refinement at the base to accurately calculate the boundary layer.



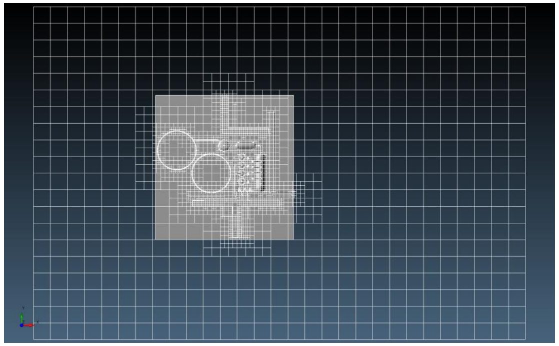
Running a Ventilation Simulation

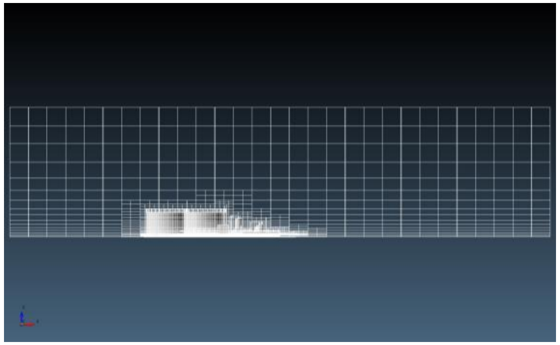
The simulation will automatically start – there is no need for any additional setup. Residuals, and flow, can be monitored as the simulation progresses.

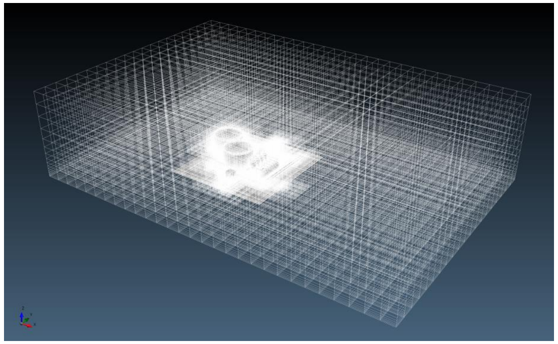


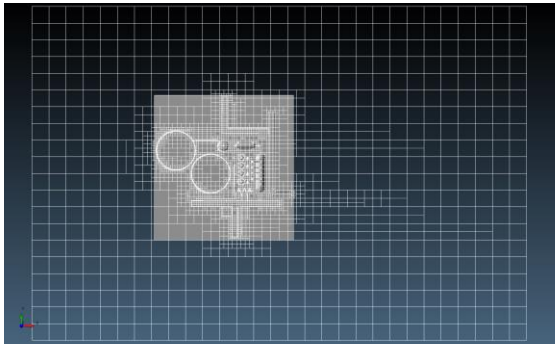
Adaptive Meshing - Ventilation

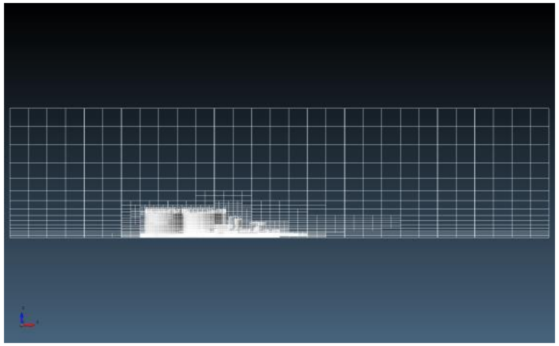
- While the simulation is running, the mesh automatically refines around regions of high gradient. Below are screenshots of before the simulation starts and after it has finished.

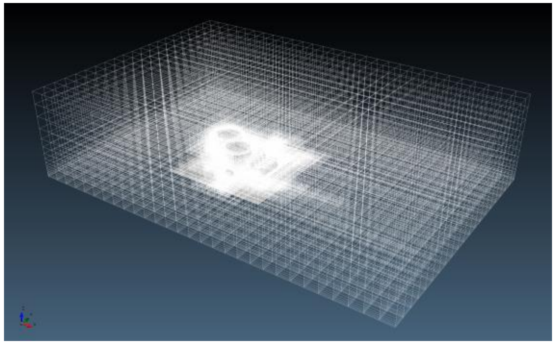








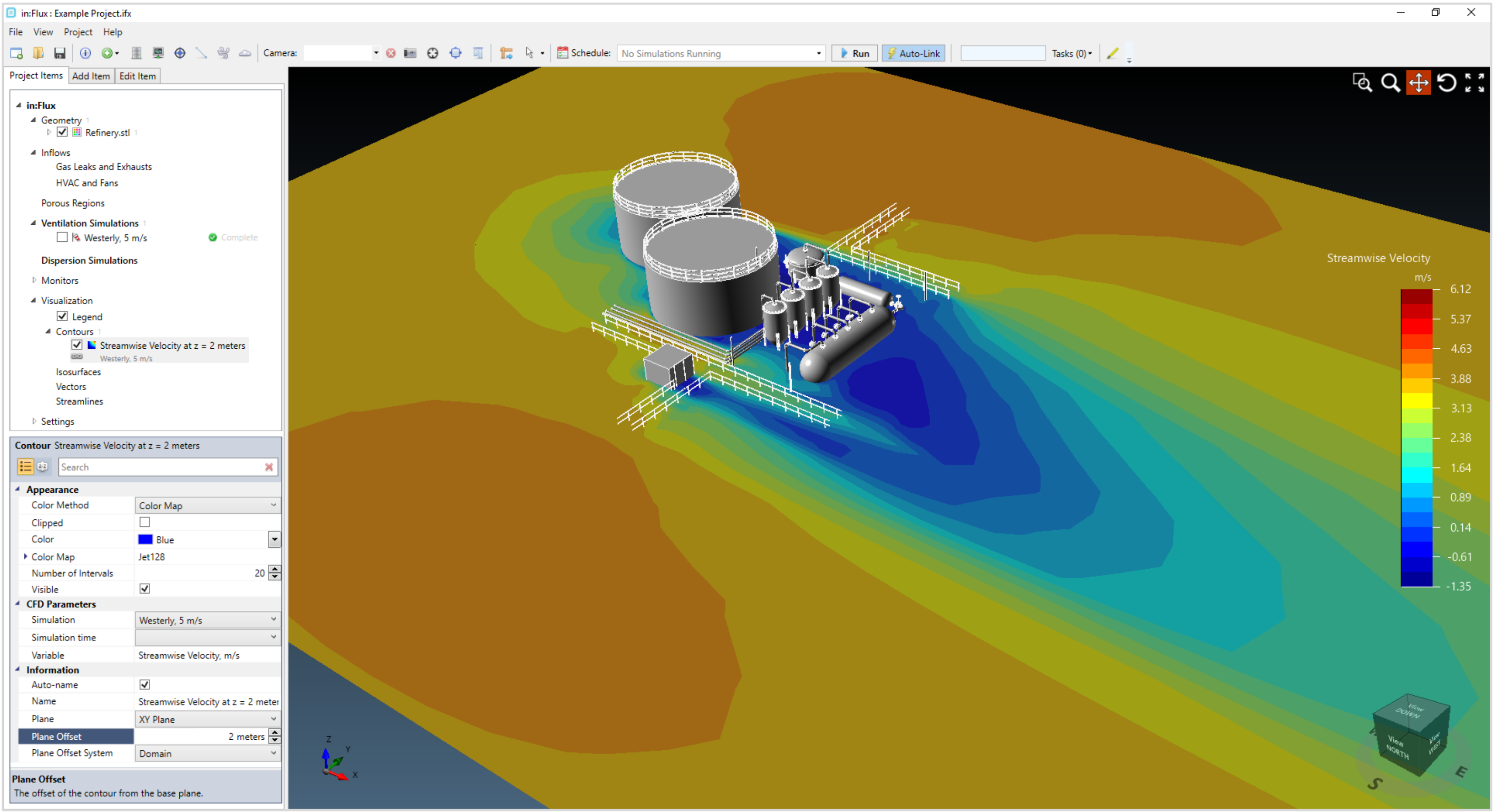




in:Flux
INTELLIGENT CFD SOFTWARE

Post-Processing: Contours

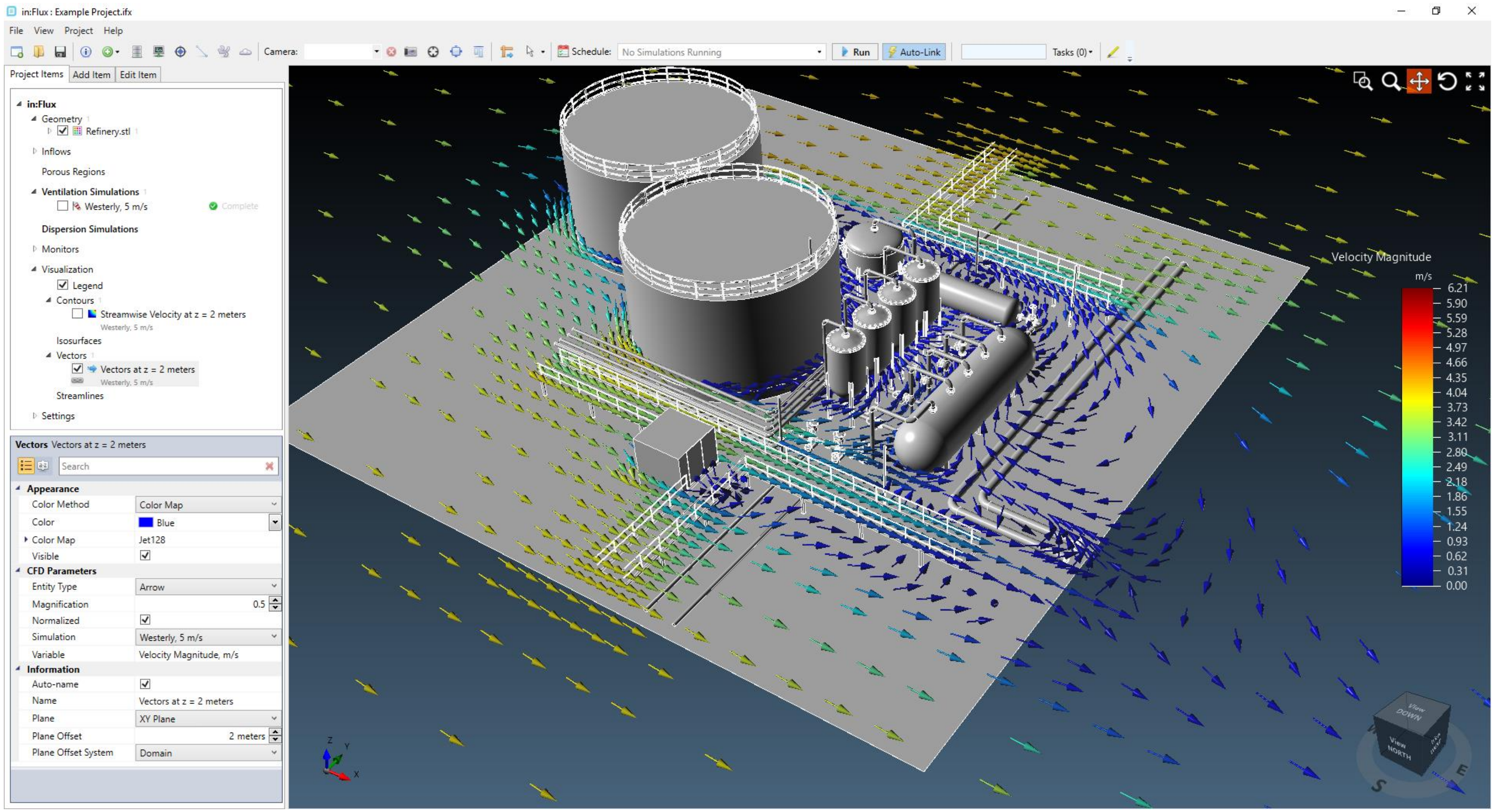
When the simulation is complete, a variety of post-processing options are available, including contours:



Developed by Insight Numerics
www.insightnumerics.com

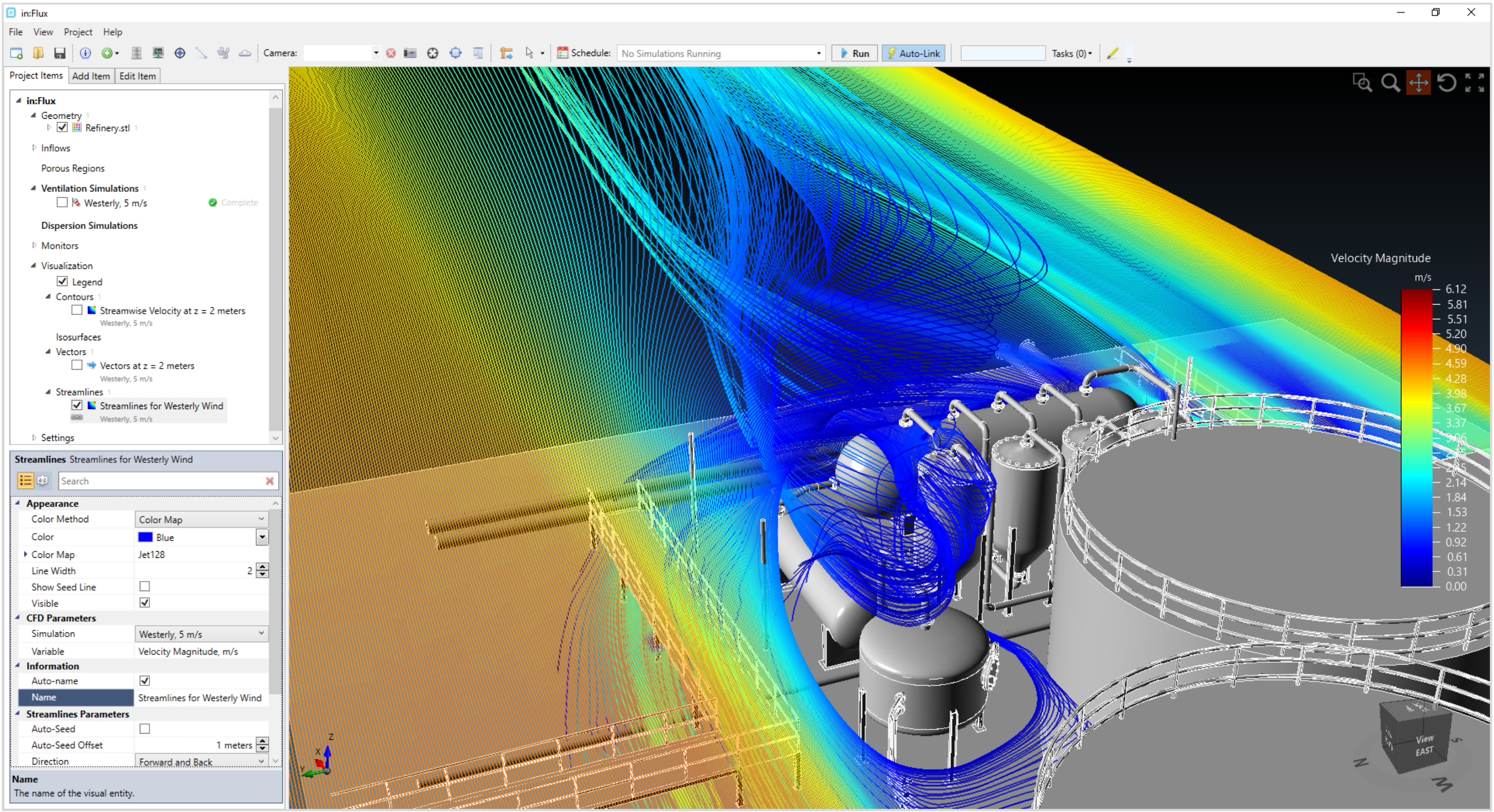
Post-Processing: Vectors

The vector field shows local flow regions around the equipment and are colored corresponding to the magnitude value.



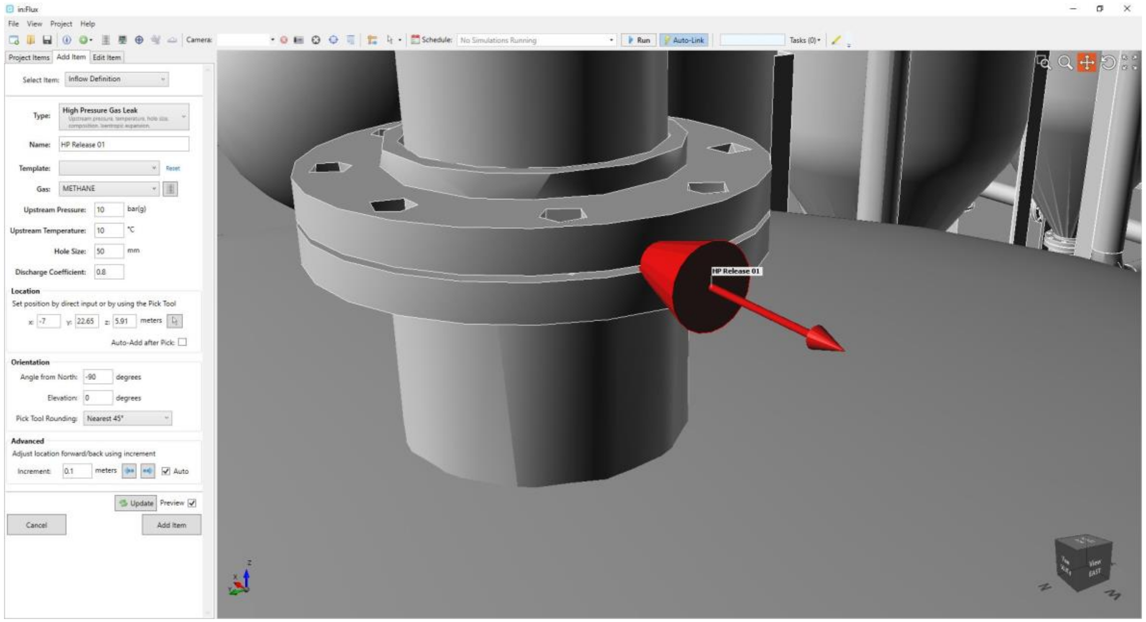
Post-Processing: Streamlines

Streamlines are useful to show circulation regions and flow through the domain:



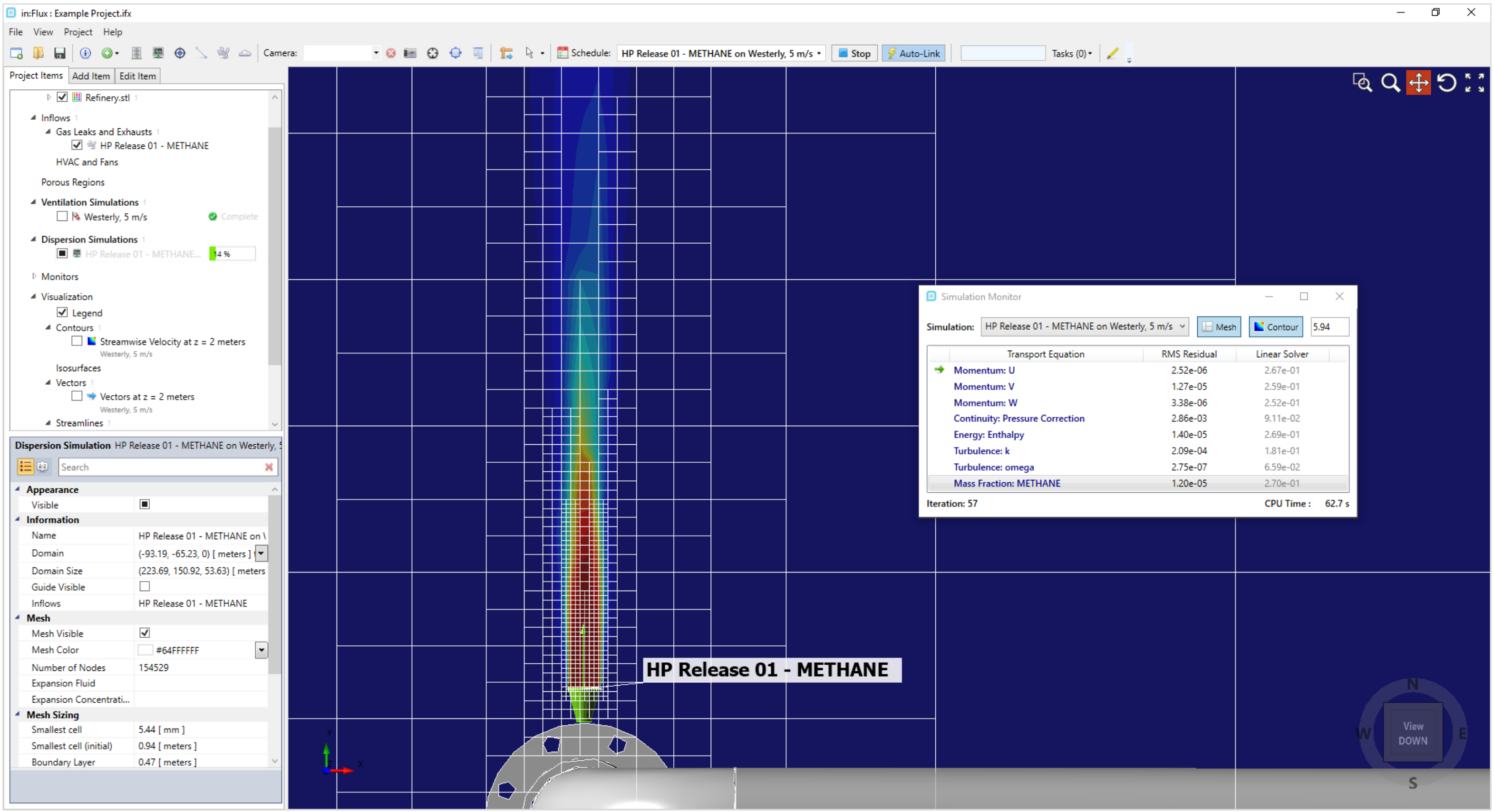
Defining Leak Sources

- Gas leaks can be defined by clicking anywhere on the CAD model and entering an upstream pressure value, temperature and composition.
- Alternatively, HVAC fans and emission sources such as exhaust plumes and can be set.
- Multi-component gases can be defined. The gas properties use the DIPPR database.



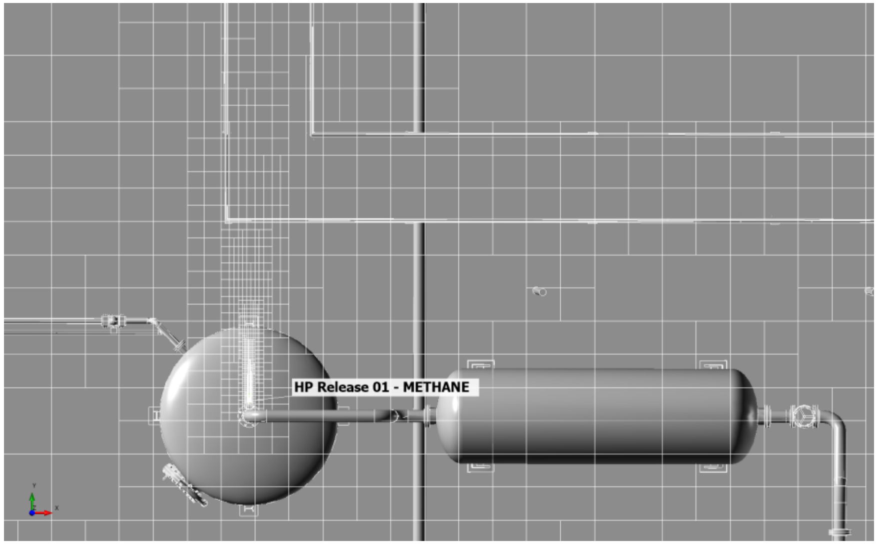
Running a Dispersion Simulation

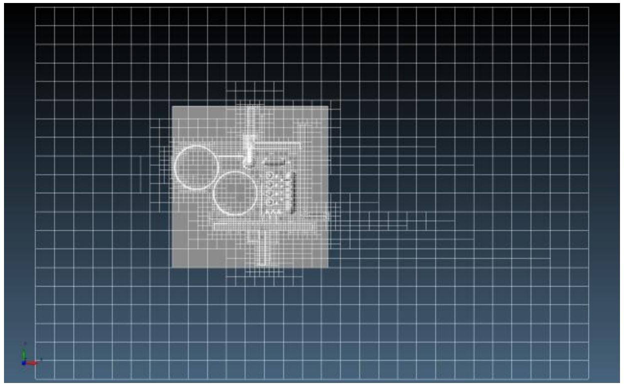
A dispersion simulation is defined simply as a combination of a leak source and a wind simulation. No further setup is required.

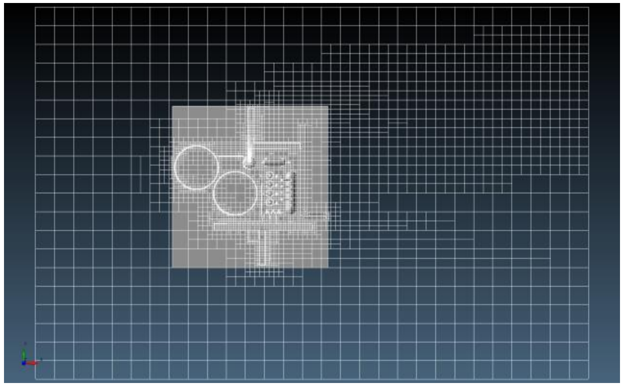


Dispersion Simulation Meshing

For dispersion simulations, the mesh adapts around regions of high concentration gradient. The mesh can be set to automatically expand to include certain concentrations. Minimum mesh size is 0.1 times the diameter of the leak.





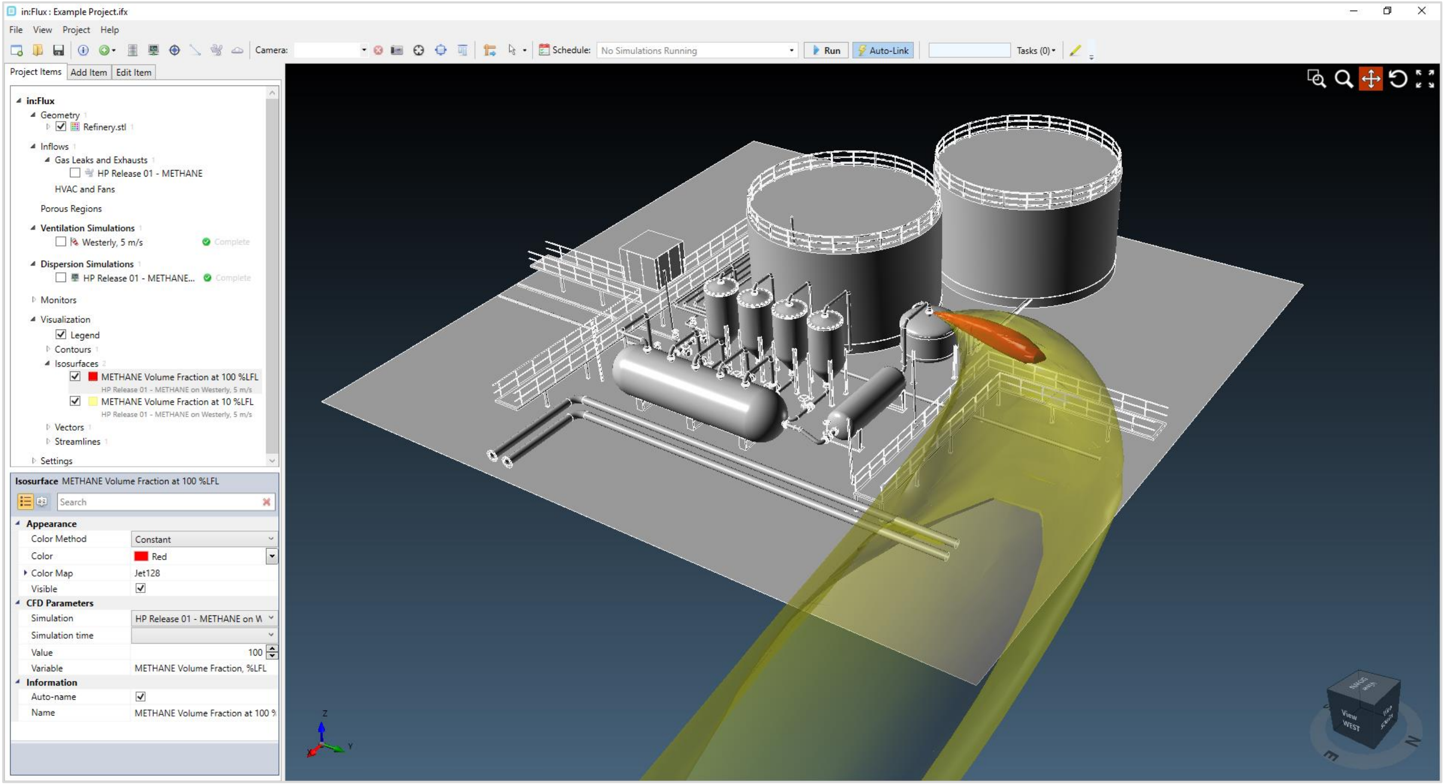


Above
Zoomed in view of mesh at leak location. The mesh will refine itself as the calculation progresses

Upper Right and Right
Top view of the auto-generated mesh at start of dispersion simulation (upper right) and after the simulation has completed (right).

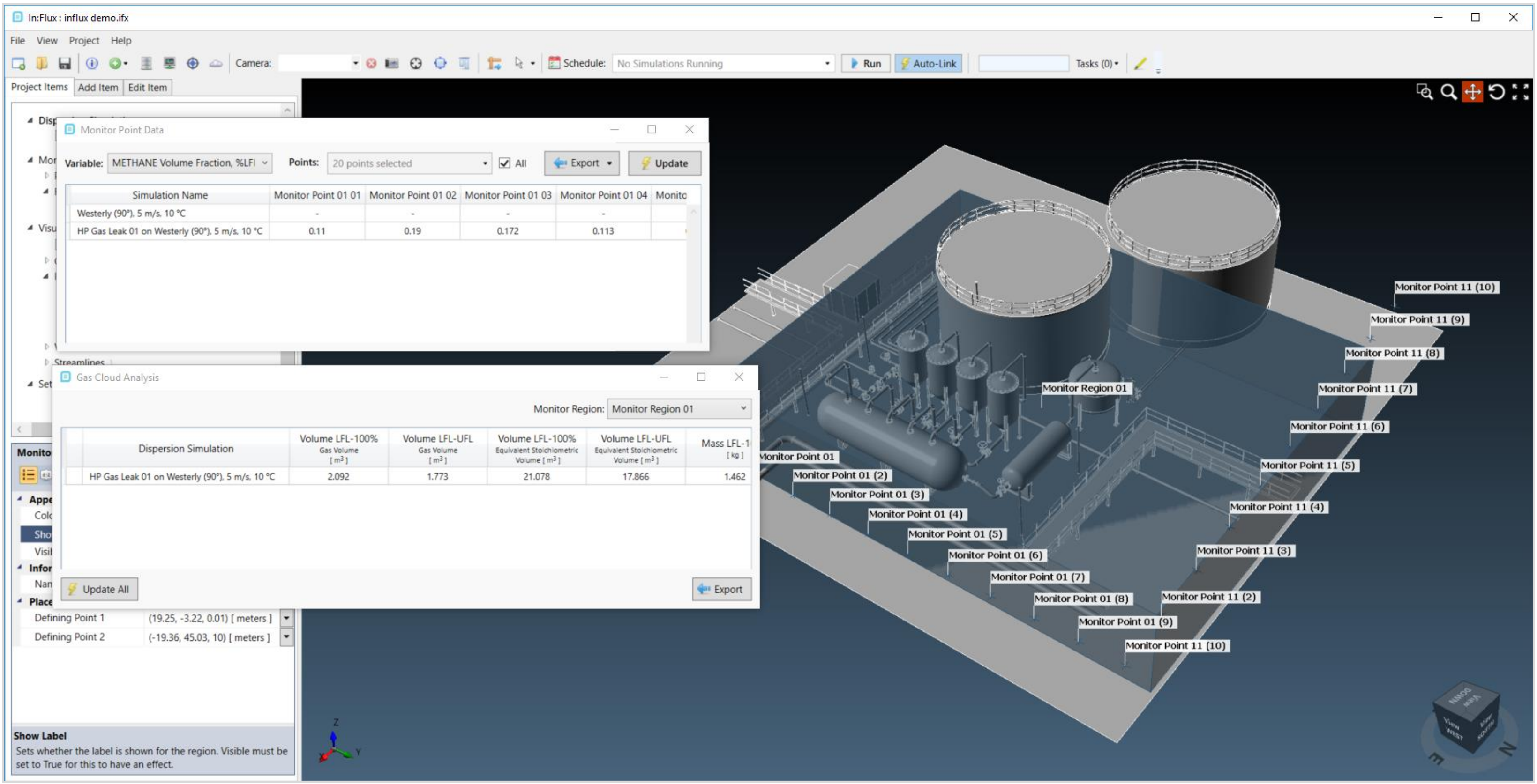
Post-Processing: Isosurfaces

Isosurfaces are useful to show the dispersed gas clouds. These can be set up as concentrations of %volume, ppm, %LEL and %UEL. The below image shows a 100% LFL cloud in red and a 10%LFL cloud in transparent yellow.



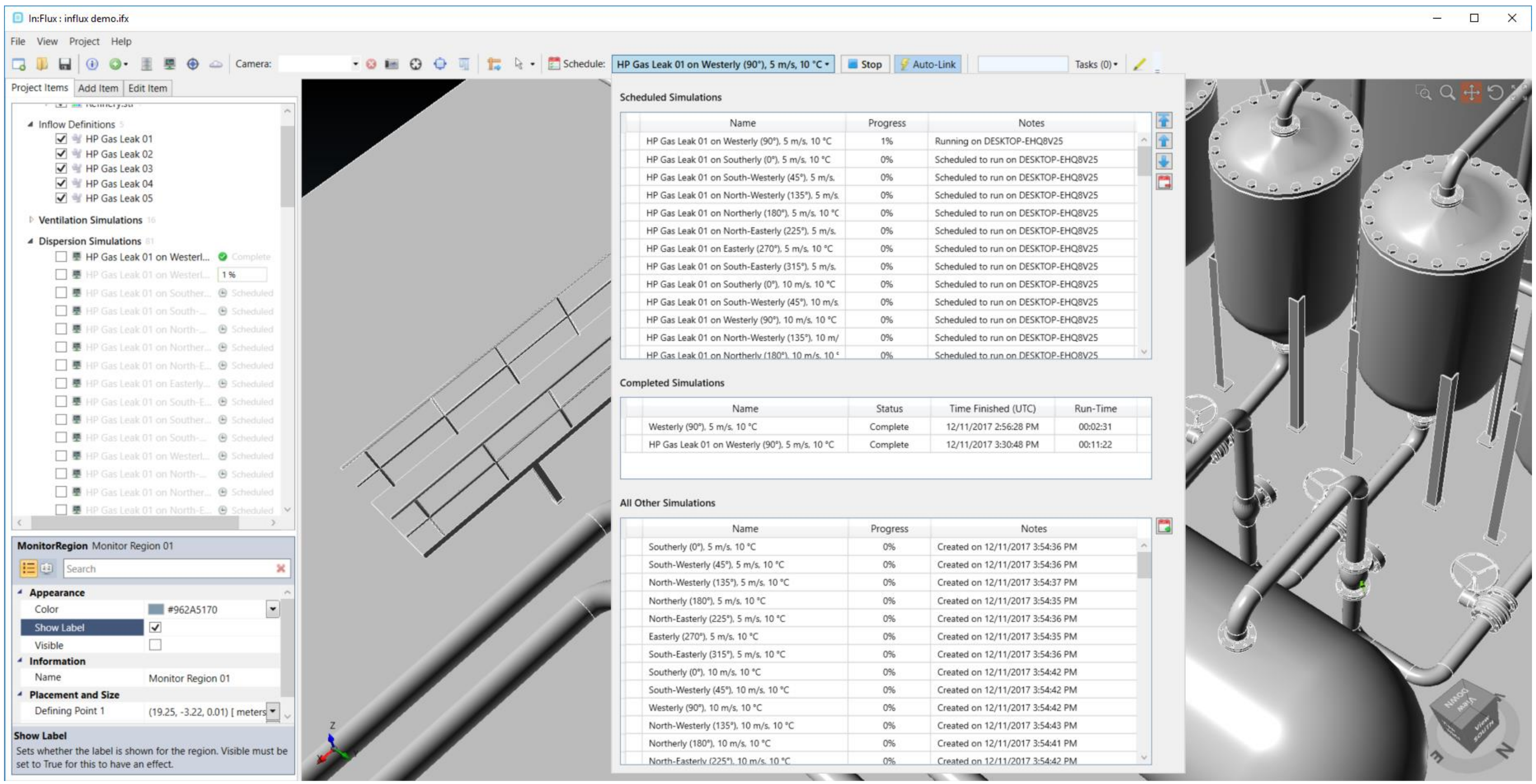
Post-Processing: Gas Monitors

Monitor points provide spot measurements of concentrations, while monitor regions can be used to calculate cloud volumes. All data is exportable to Excel. There is no limit to the number of monitors in an in:Flux project.



Multi-Run Project

The example simulation in this presentation has shown one ventilation and one dispersion. However, with just a few clicks, it is easy to setup over 100 simulations. Multiple simulations will run sequentially using the simulation scheduler, shown below, without the need of scripting.

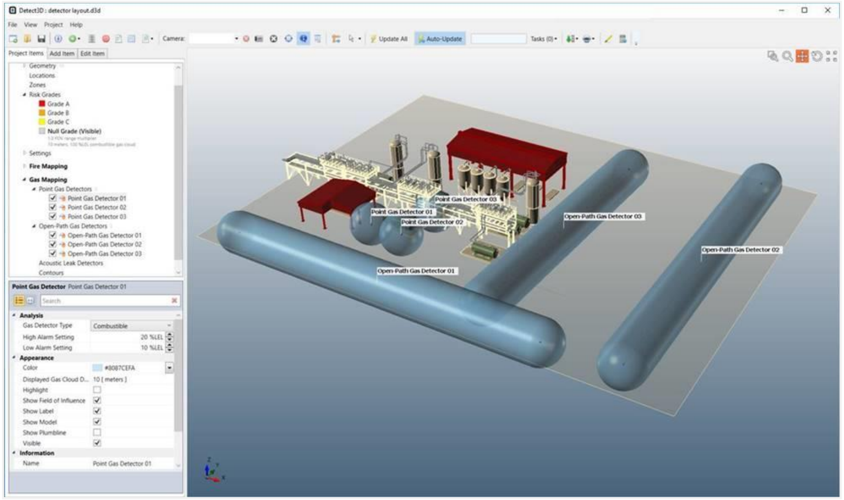


in:Flux INTELLIGENT CFD SOFTWARE

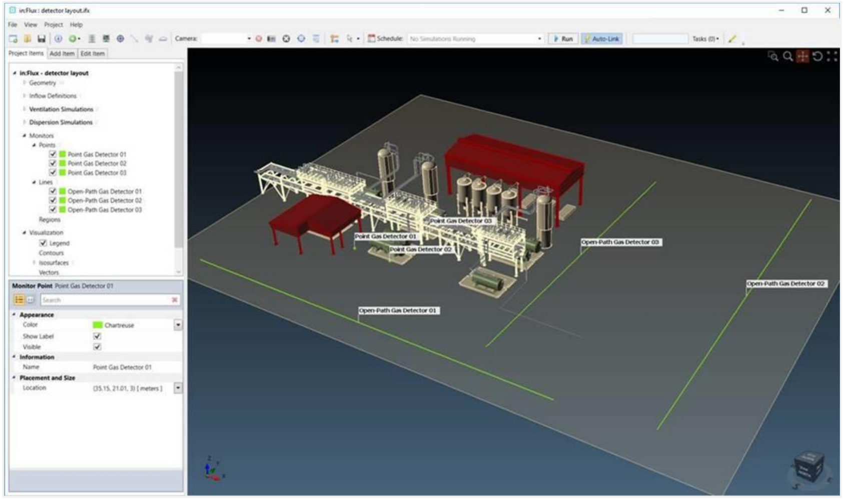
Developed by Insight Numerics
www.insightnumerics.com

Detect3D and in:Flux Integration

- Once a CAD file has been loaded in either software, it can be transferred without the need to import the original CAD file
- For scenario gas mapping, point and open path gas detectors can be imported from Detect3D to in:Flux as monitor point and monitor lines to received %LFL and LEL.m data



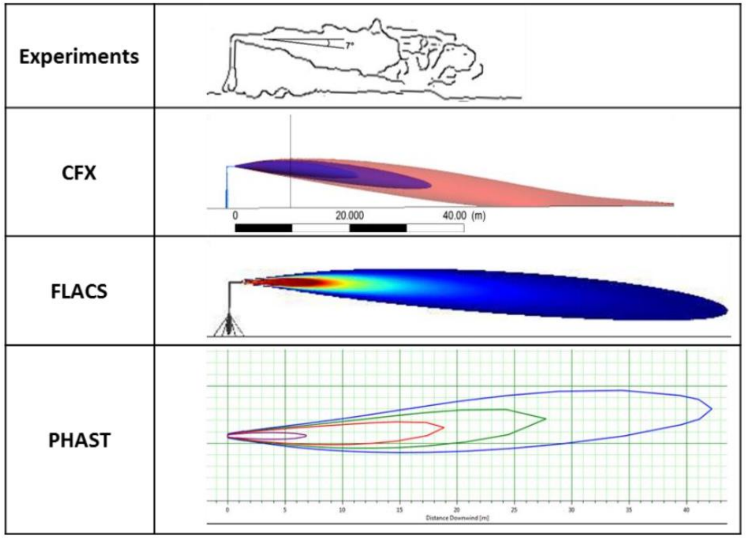
Detect3D – CAD model, point and open path gas detectors positioned in project



in:Flux – CAD model imported from Detect3D as well as the gas detectors. Detectors now show up as monitor points and monitor lines, shown in yellow above.

Validation

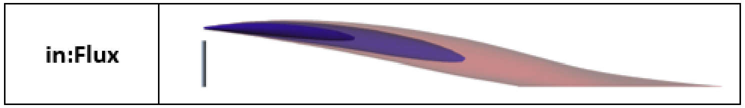
- in:Flux has been compared to known data for jet leaks, and performs as well as other CFD software such as Ansys-CFX and OpenFOAM.
- A variety of experimental data has been used to validate in:Flux, and it has performed as well, or better, than other CFD software.



Above - Experimental rig for LNG vapor release (18” vertical plume shown), Ref [1].

Top Right - Plume paths for various models tested in Ref [1], horizontal 6” diameter release.

Bottom Right – Results from . Plume path for in:Flux, horizontal 6” diameter release.



Ref [1] - Quillatre, P. Relevance of the current modeling methods for the prediction of LNG vapor dispersion and development to be carried on. IChemE Symposium Series No. 162, Hazards 27 Conference, 2017.

in:Flux Technical Details

- in:Flux solves the 3D Reynolds-Averaged Navier Stokes (RANS) equations.
- The RANS equations are closed using the SST turbulence model with standard wall functions.
- Pressure-velocity coupling uses the SIMPLE algorithm.
- Buoyancy effects due composition and temperature are included using a full buoyancy model.
- Numerical discretization is second-order, using second-order upwind for convection and central differencing for diffusion.
- The Immersed Boundary Method (IBM) is used to account for geometry. Porous regions are not used.

Recent and Future Developments

- As of January 2019, in:Flux now comes included with:
  - Navisworks (NWD) import via .dwf files
  - Risk-based CFD analysis
  - Gas Detector Optimization Capability using CFD
  - Internal Ventilation capabilities (including HVAC and flow-through fans)
  - Transient Simulations, including shut-down
- Future developments planned for in:Flux include:
  - Multi-phase leaks (sprays, rain-out etc.) and associated physics.
  - Modeling for jet and pool fires
  - Much more…
- The long term goal for Insight Numerics is to develop an explosion capability within in:Flux.
- As the sole developers of in:Flux, Insight welcomes feedback from its users to further develop the software.

Next Steps...

- in:Flux can be downloaded by registering on

  www.insightnumerics.com

- Upon registration, approved users receive a free 10-day evaluation license which give access to tutorials, user guides, and example CAD files

- Check out our YouTube Channel for how-to videos and descriptions

- To hear about Insight Numerics and our products' capabilities, sign up for our newsletter or contacts us at

  info@insightnumerics.com