

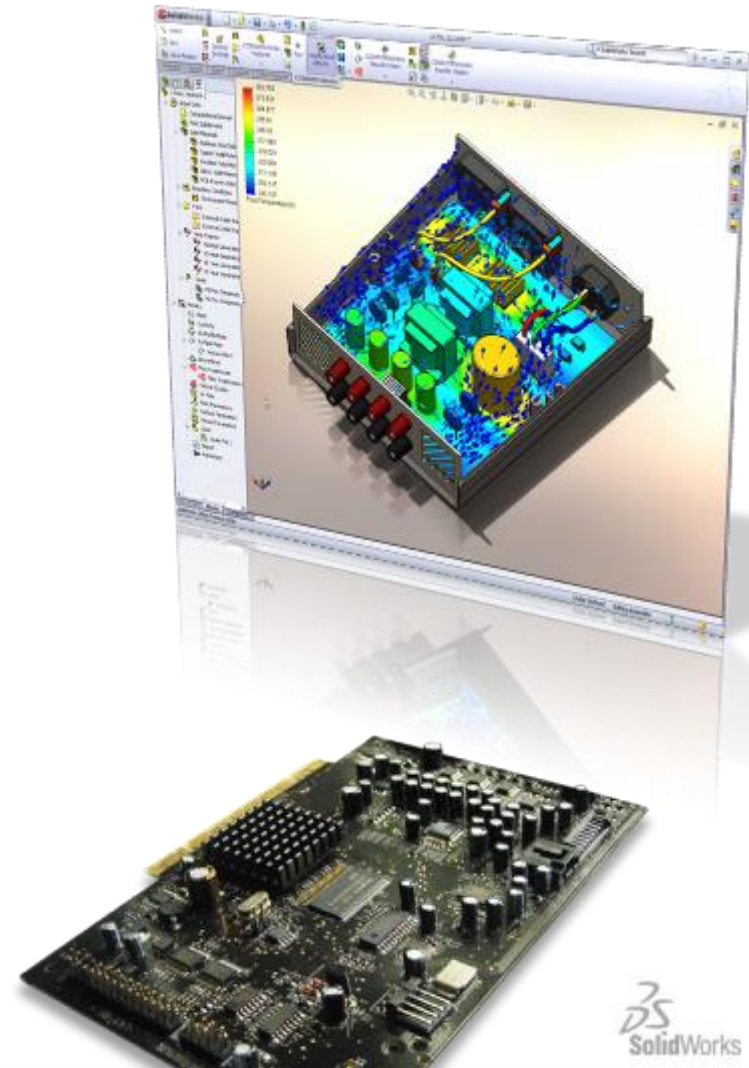
A full-page background image of a snowboarder in an orange jacket and dark pants, captured mid-air while performing a jump on a snowy mountain. The snowboarder is leaving a trail of white snow behind them. The background shows a vast, snow-covered mountain range under a clear blue sky.

real inspiration

SolidWorks FloXpress

Agenda

- What is FloXpress and how do you use it?
- Case study – Internal Analysis
 - Venturi
- How does it work?
- Case studies – Internal Analysis
 - Water Tank, Electrical Casing
- Limitations
- Exercise 1 - Valve – various opening angles
- Exercise 2 - Regulator – with or without bores
- Case study – External Analysis
 - HGV in a Wind Tunnel
- Exercise 3 - Wing mirror design comparison
- Tips & Tricks
- Going further with Flow Simulation

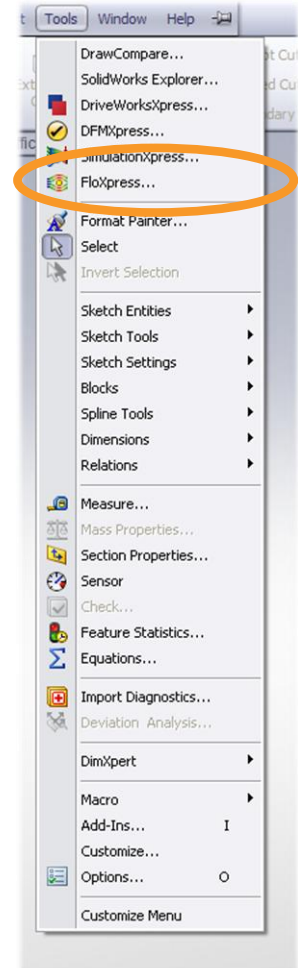
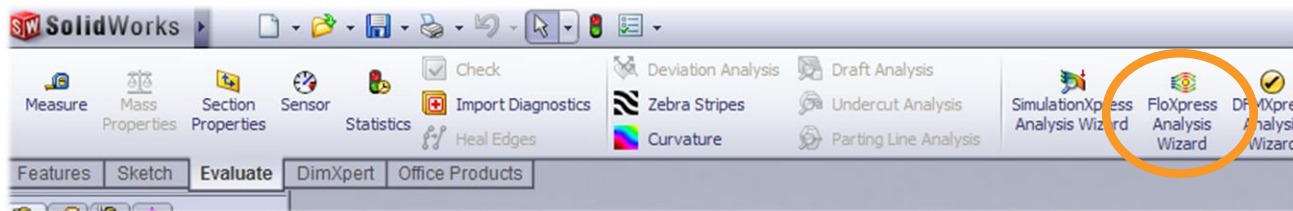


What is FloXpress?

- A tool simulating fluid flow
- Available to all SolidWorks users from SW2008 onwards..
- A 'first pass' tool for simple flow problems
- Very easy to use for non-fluids experts
- It helps you:
 - Understand how fluid is likely to flow through a closed space
 - See where there are regions of high flow
 - Determine the maximum speed of flow
 - See where there are regions of flow stagnation (potential 'hot spots')
 - Compare different geometries to assess how flow is affected so you can choose the best design

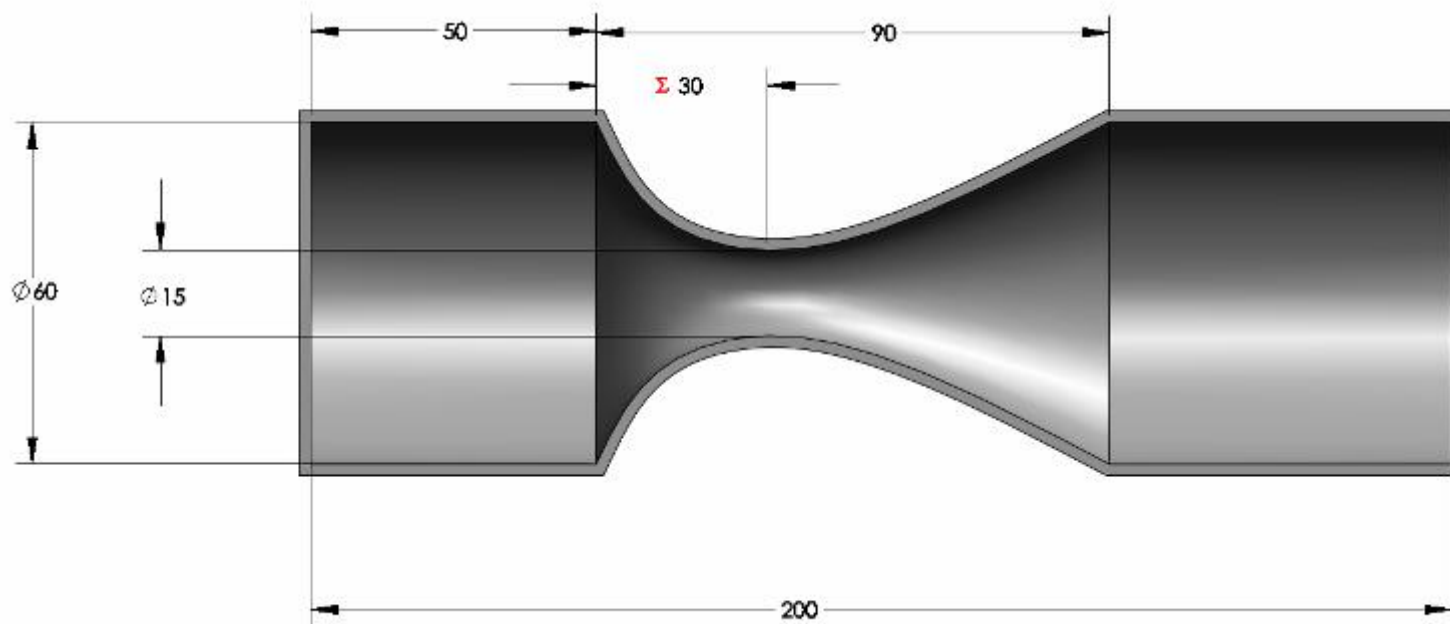
How do you use it?

- Before you start:
 - Simplify the part / assembly wherever feasible
 - Check for the smallest flow passage (using the measure tool)
 - Close off any openings by creating 'lids'
- To start FloXpress:
 - Tools > FloXpress
 - Evaluate Command Manager tab

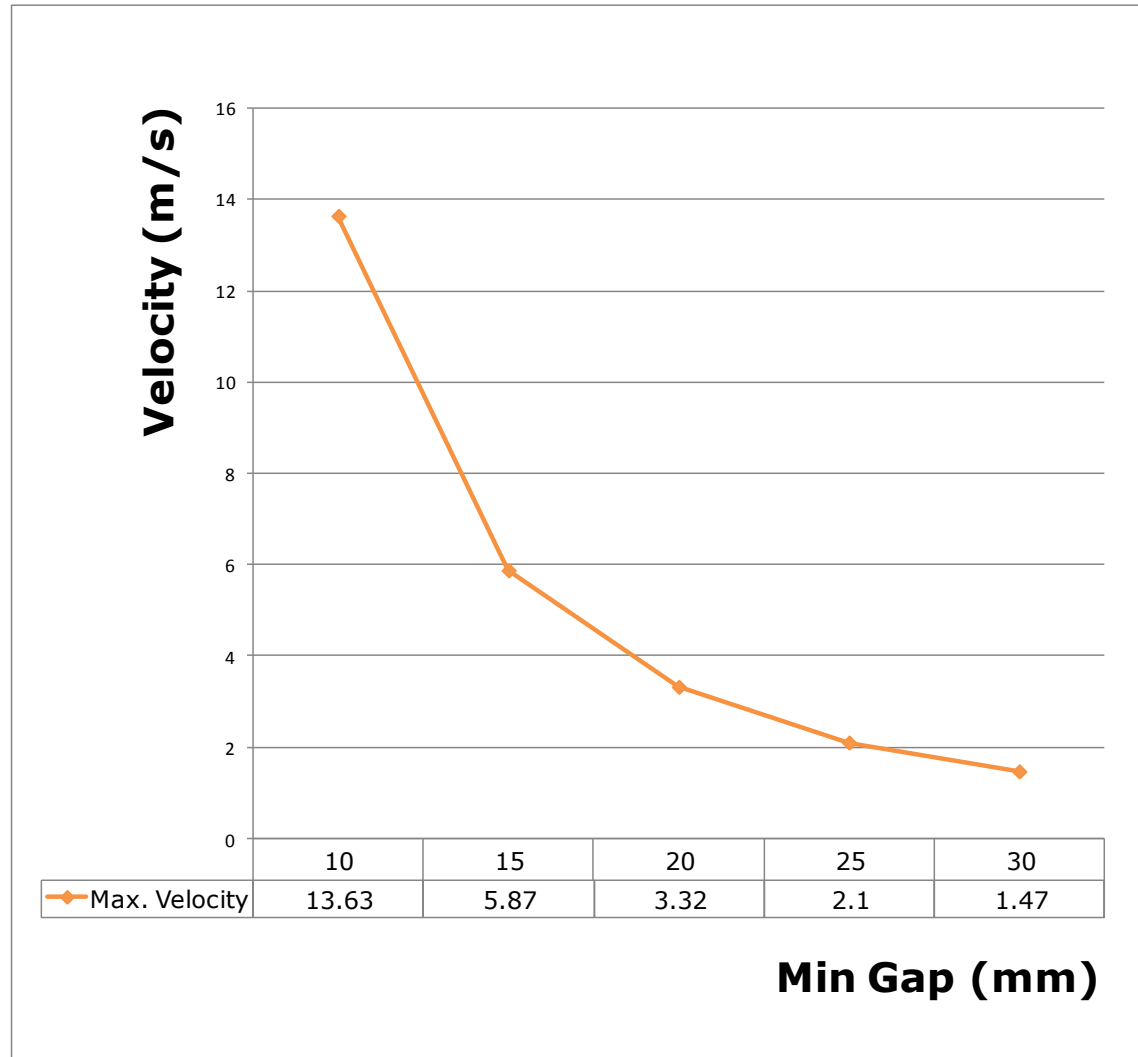


Case Study - Venturi

- What is the maximum flow speed?
- How does flow vary with minimum gap size?

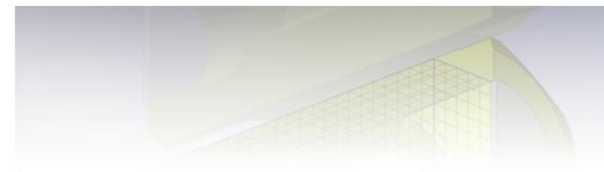
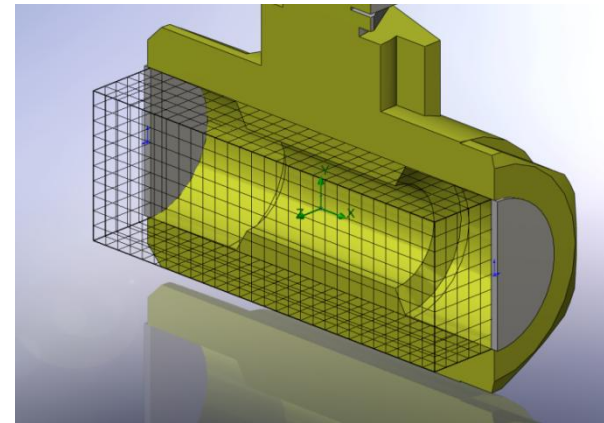


Venturi Results

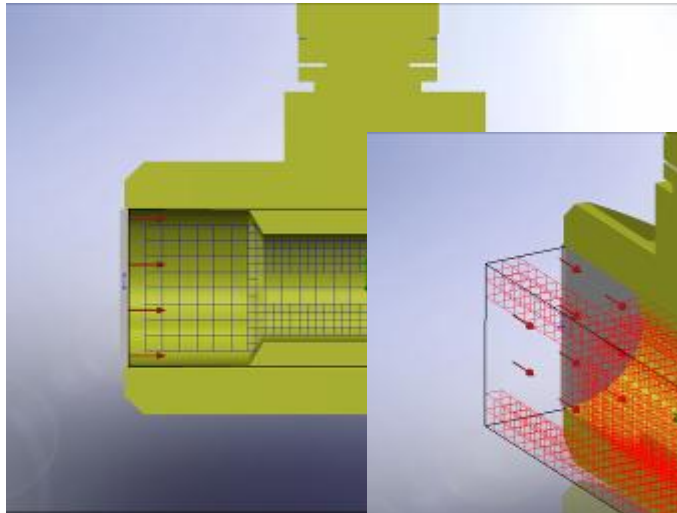


How does it work?

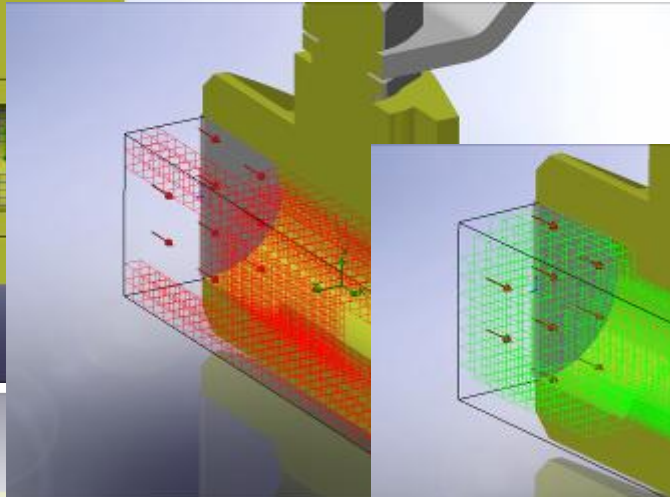
- Uses the 'Finite Volume' method ('FV')
- The region of analysis is a rectangular 'volume' broken into smaller rectangular prisms
- The region includes both solids and fluids
- The region is called the 'computational domain'
- Each prism is called a 'cell'
- The solver seeks to find a solution that satisfies all of the fluid equations



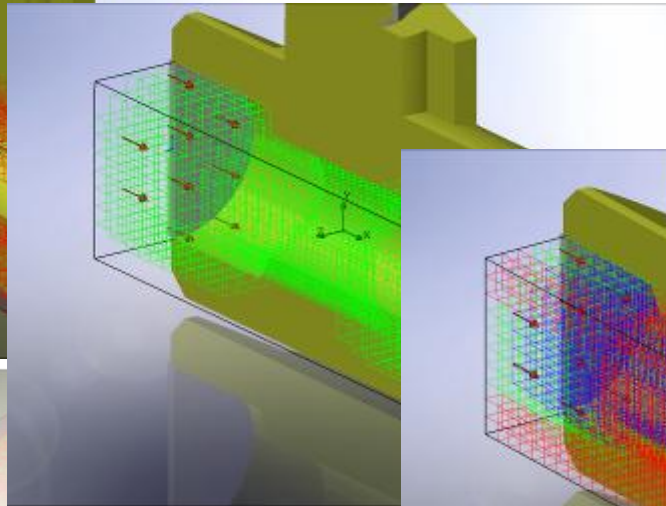
How does it work?



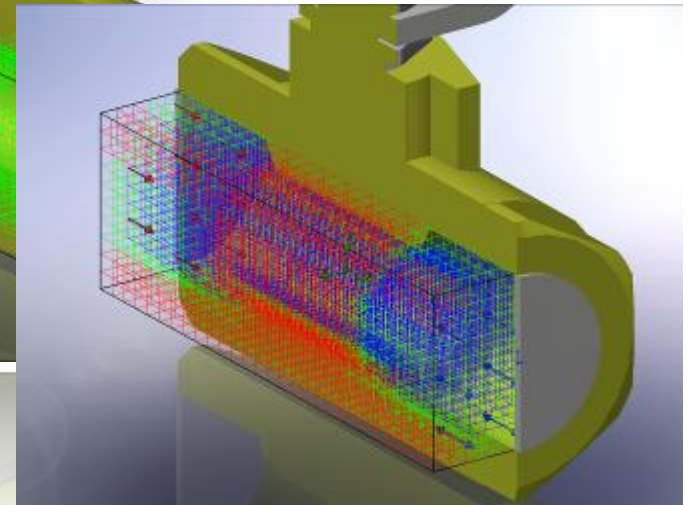
Fluid cells



Solid cells



Partial cells



All cells

How does it work?

- The solver uses the following principles:
 - Conservation of mass
 - Conservation of momentum
 - Conservation of energy
 - Fluid behaviour as described by Navier Stokes

- The solver is 'iterative' (as unknowns > knowns)
 - It 'guesses' a set of values for each cell (pressure, temp, velocity etc)
 - Inserts the values into the equations
 - Determines the error (imbalance) in the equations
 - Makes an intelligent guess at the next set of values
 - Repeats until the error reaches an acceptable 'stopping criteria'

How does it work?

The conservation laws for mass, angular momentum and energy in the Cartesian coordinate system rotating with angular velocity Ω about an axis passing through the coordinate system's origin can be written in the conservation form as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1.1)$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) + \frac{\partial p}{\partial x_i} = \frac{\partial}{\partial x_j}(\tau_{ij} + \tau_{ij}^R) + S_i \quad i=1,2,3 \quad (1.2)$$

$$\frac{\partial \rho H}{\partial t} + \frac{\partial \rho u_i H}{\partial x_i} = \frac{\partial}{\partial x_i}(\mu_j(\tau_{ij} + \tau_{ij}^R) + q_i) + \frac{\partial p}{\partial t} - \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + \rho \varepsilon + S_i u_i + Q_H, \quad (1.3)$$

$$H = h + \frac{u^2}{2},$$

where u is the fluid velocity, ρ is the fluid density, S_i is a mass-distributed external force per unit mass due to a porous media resistance ($S_{i_{porous}}$), a buoyancy ($S_{i_{gravity}} = -\rho g_i$, where g_i is the gravitational acceleration component along the i -th coordinate direction), and the coordinate system's rotation ($S_{i_{rotation}}$), i.e., $S_i = S_{i_{porous}} + S_{i_{gravity}} + S_{i_{rotation}}$, h is the thermal enthalpy, Q_H is a heat source or sink per unit volume, τ_{ijk} is the viscous shear stress tensor, q_i is the diffusive heat flux. The subscripts are used to denote summation over the three coordinate directions.

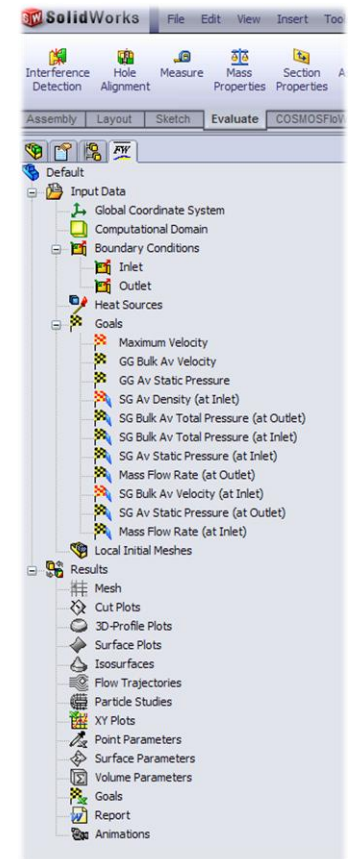
Theoretical Background

- Due to the FV method, the governing equations are discretized in a conservative form.
 - The spatial derivatives are approximated with implicit difference operators of second-order accuracy.
 - The time derivatives are approximated with an implicit first-order Euler scheme.
 - The viscosity of the numerical scheme is negligible with respect to the fluid viscosity.
- The second-order upwind approximations of fluxes are based on the implicitly treated modified Leonard's QUICK approximations and the Total Variation Diminishing(TVD) method.

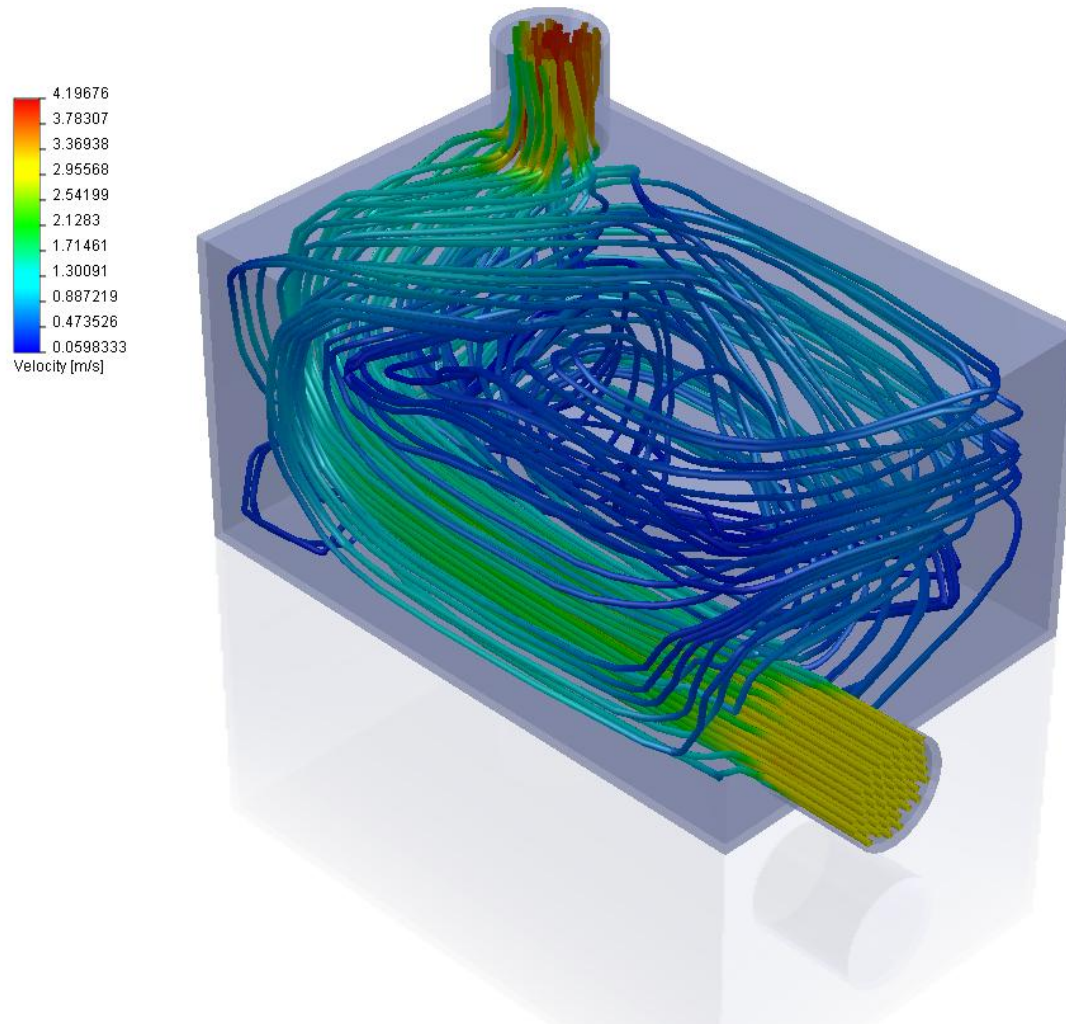


How does it work?

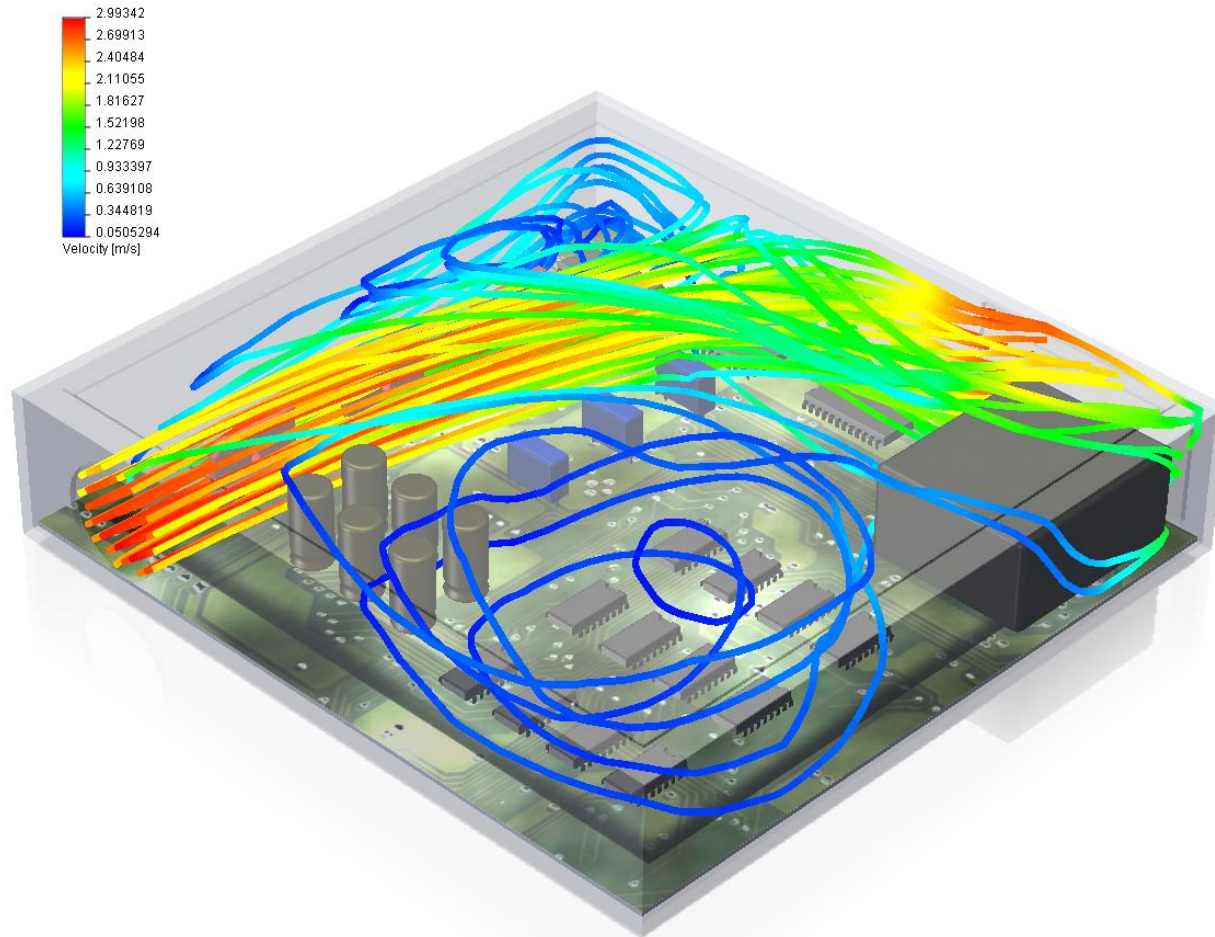
- It uses the full Flow Simulation solver (not an inferior version)
- It is wizard driven – to make it easy!
- The wizard drives the inputs to a Flow Simulation analysis
 - 2 inputs ('Inlet' and 'Outlet')
 - 11 'Goals'
- FloXpress simulations can be used in full Flow Simulation



Case Study - Water Tank

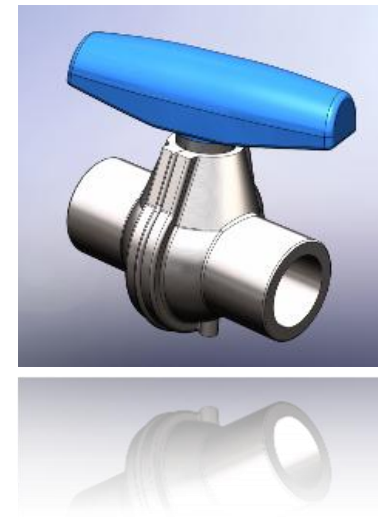
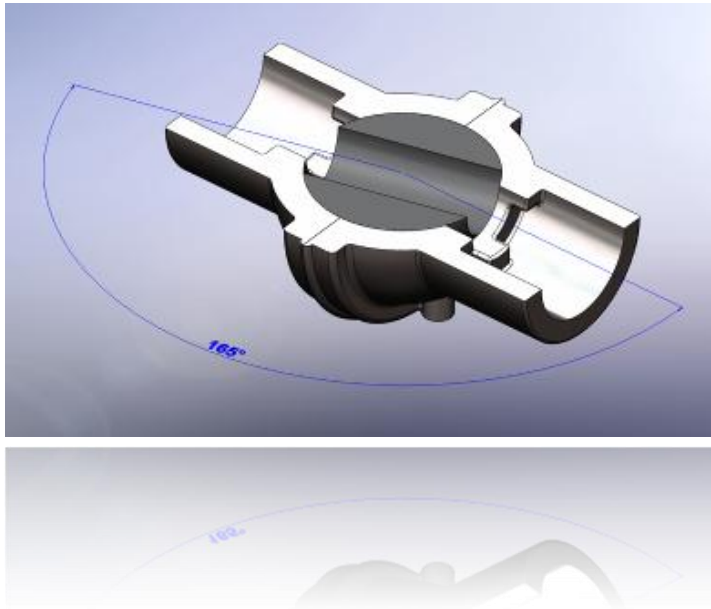


Case Study – Electrical Casing



Exercise 1 - Ball Valve

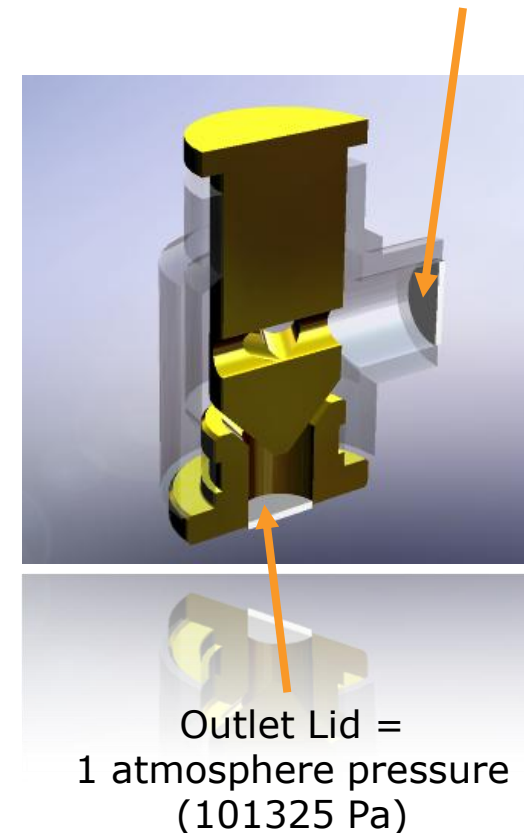
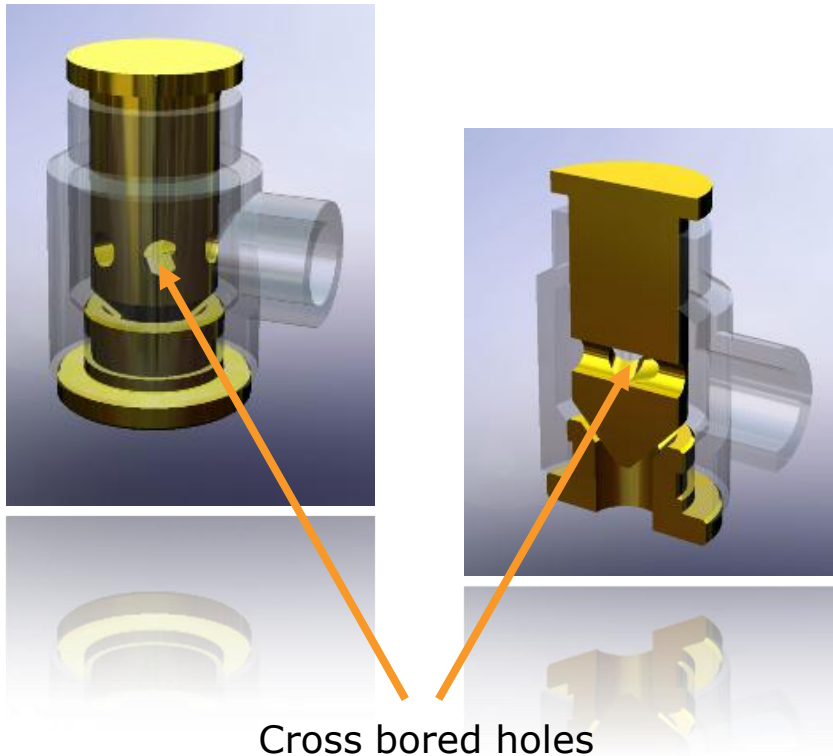
- Create inlet and outlet lids
- Inlet = 0.5kg/sec mass flow rate
- Outlet = 1 atmosphere (101325 Pa)
- What is the maximum velocity?
- What is affect of changing the opening angle? (tip: edit the angle mate)



Exercise 2 - Regulator

- Fluid is water
- How do the cross bore holes in the 'Conical Adjuster' part affect the flow? (tip: change the configuration)

Inlet Lid =
2 kg/sec
mass flow rate

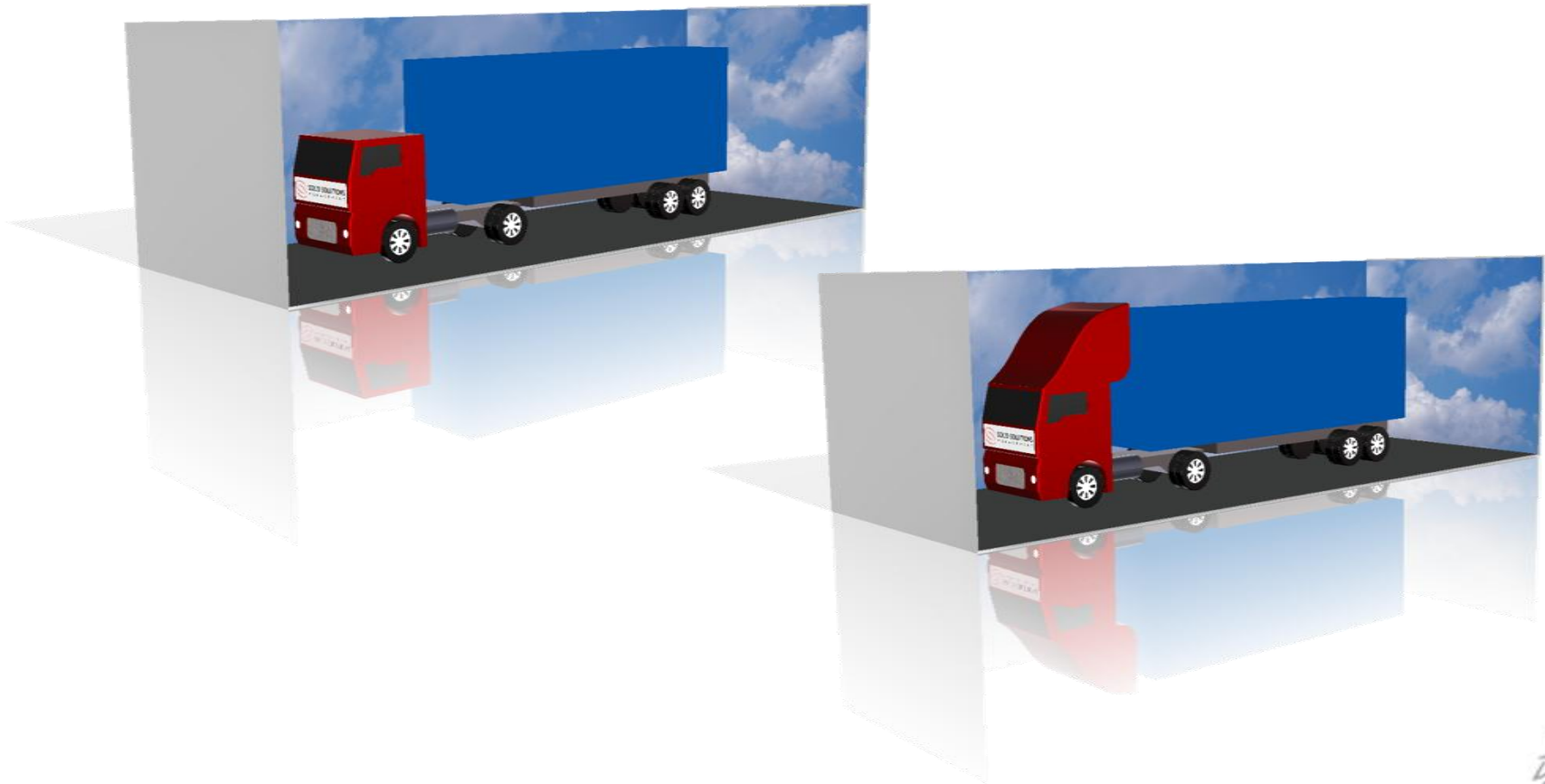


Important limitations

- Designed for Internal Analysis only (but can use a wind tunnel)
- Cannot use surfaces, only solids (including lids)
- Can simulate with only 2 fluids – air, water
- No thermal inputs (fluid temperature is calculated but not available for plots)
- Limited quantitative output (only maximum velocity)

Case Study 2 - HGV in a Wind Tunnel

- Without a cab deflector
- With a cab deflector



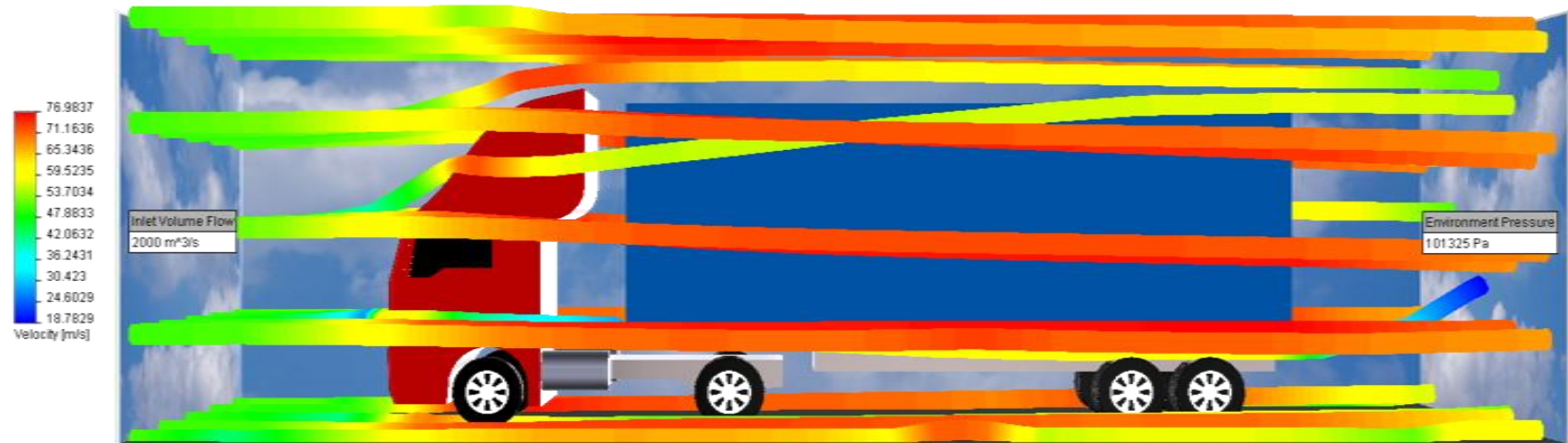
Case Study 2 - HGV in a Wind Tunnel

- External analysis
 - Create a 'wind tunnel' to surround the assembly
 - Allow enough space for the flow to develop
 - Set inlet flow as an inlet volumetric flow rate
 - Truck is travelling at 55mph
 - 1 m/s = 2.24 mph
 - Multiply the velocity in m/s by inlet area to get flow rate
 - $(55/2.24) \times (6 \times 7) = 1033\text{m}^3/\text{s}$

- Note: FloXpress cannot simulate 'ground effect'

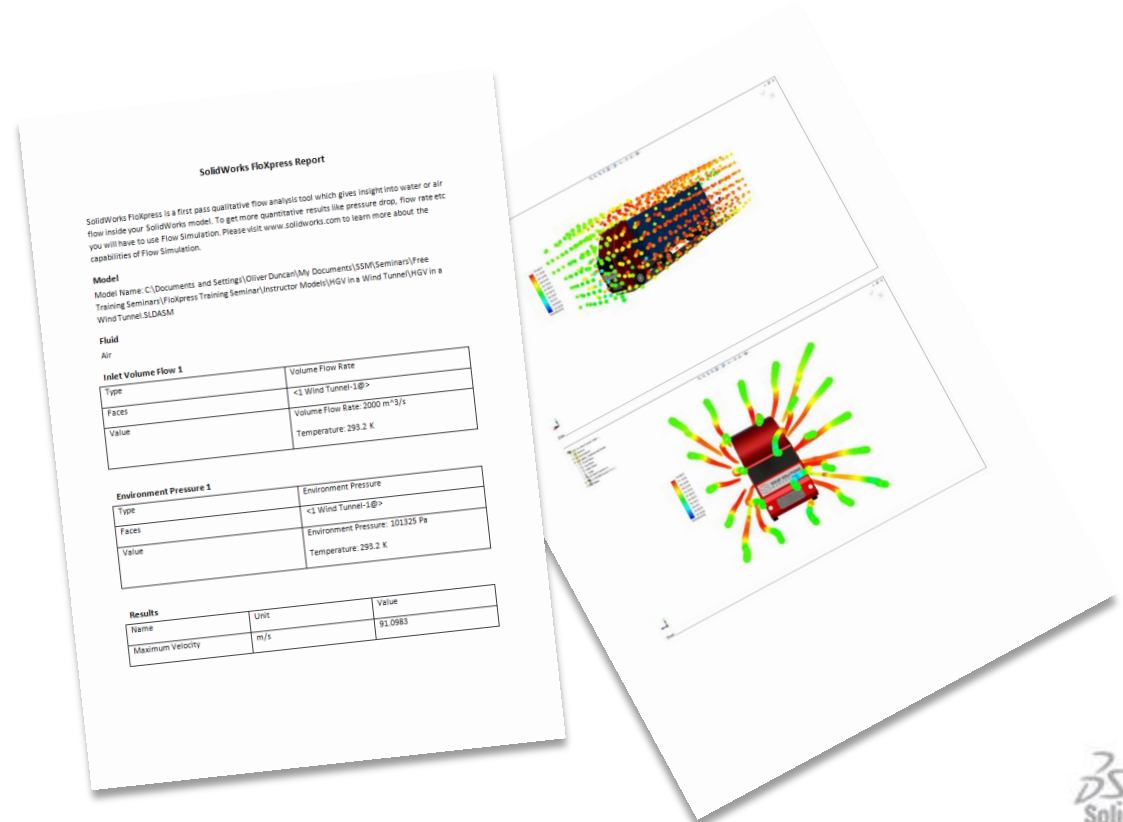
Convert from	Multiply by						
	Convert to						
	m/s	km/h	ft/min	ft/s	yards/min	mph	knots
m/s	1	3.6	196.9	3.28	65.6	2.24	1.94
km/h	0.28	1	59.0	0.91	18.3	0.62	0.54
ft/min	0.005	0.018	1	0.017	0.033	0.011	0.009
ft/s	0.30	1.1	60	1	2.0	0.68	0.59
yards/min	0.016	0.055	30	0.055	1	0.34	0.29
mph	0.45	1.6	1056	1.47	336	1	0.87
knots	0.51	1.9	1152	1.68	370	1.15	1

Case Study 2 - Results



Case Study 2 - Results

- Report
 - Word document
 - Pictures of 'snapped' images can be auto-inserted into report or pasted manually from .fxp folder



Exercise 3 – Wing mirror

- External analysis
- Compare wing mirror designs (tip: change the configuration)
- Flat vs Aerodynamic at around 70mph (multiply the velocity in m/s by inlet area to get flow rate)
- Use volumetric flow rate

1 degree Celsius
= 274.15 kelvin

1 atmosphere
= 101325 pascals



Convert from	Multiply by						
	Convert to						
m/s	m/s	km/h	ft/min	ft/s	yards/min	mph	knots
1	1	3.6	196.9	3.28	65.6	2.24	1.94
km/h	0.278	1	10.9	0.91	0.617	0.617	0.539
ft/min	0.0167	0.0167	1	0.0167	0.0167	0.0167	0.0147
ft/s	0.305	0.305	0.305	1	1.09	1.09	0.914
yards/min	0.0059	0.0059	0.0059	0.0059	1	1	0.87
mph	0.447	0.447	0.447	0.447	0.447	1	0.868
knots	0.514	0.514	0.514	0.514	0.514	0.514	1

Tips and Tricks

- Use section view to help select internal faces
- Use virtual in-context modelling as 'lids' in assemblies
- Be careful with sizes of narrow gaps
- Parts can be hidden (rmb on part)
- Images are saved in a 'fxp' sub folder
- Use Sketch Pictures or Decals for backgrounds
- 2009 enhancements:
 - Display annotations
 - Preview of saved images



Tips and Tricks

- Clicking the 'next' arrow at the Welcome page performs 'Check Geometry'
- Use 'View fluid volume' to ensure that the correct volume has been identified
- Solver bar may go backwards as well as forwards!
- Technical details are in .stdout file (under fxpn folder)
- FloXpress studies and results can be carried forward to Flow Simulation
- Allow enough upstream flow for the fully developed flow to be created
- You can record the animation using SolidWorks Rx – edit in MS Movie Maker
- PDF of validation examples and tutorials can be supplied, please ask

Going further with Flow Simulation

- FloXpress - much simplified version of SolidWorks Flow Simulation

- Full Flow Simulation capabilities:
 - Internal or external analyses
 - Wide choice of liquids and gases
 - Full thermal heat transfer capability (include solar radiation)
 - Buoyancy calculations for natural convection problems
 - Multiple fluid sub-domains
 - Heat sources, fans, heat sinks, TEC's
 - Multiple inlets and outlets
 - Automatic and manual mesh controls; adaptive refinement
 - Porous media
 - Real time solver monitoring
 - Extensive results outputs
 - Automatic export of results to Excel and graphing