Introducing a highly efficient CFD solver for low-speed fan analysis

Wout PONCELET – **Cadence Design Systems**

Domenico MENDICINO – **Cadence Design Systems**

27 – 29 June 2022, Senlis (France)

Cadence Design Systems

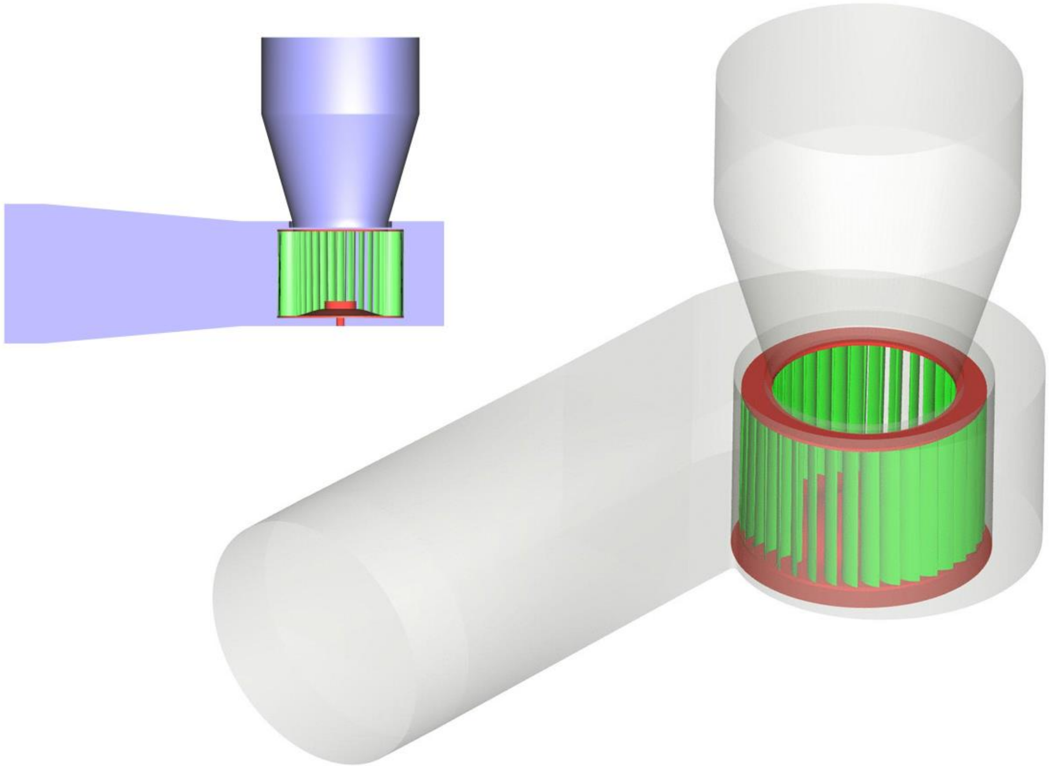
- **Cadence Design Systems** is a pivotal leader in electronic design, building upon more than 30 years of computational software expertise.
- In 2021, Cadence acquired **NUMECA International** expanding system analysis capabilities with Computational Fluid Dynamics (CFD)
- Leadership in the simulation of fluid flows and heat transfers, covering the entire simulation chain:
  - CAD treatment
  - Mesh generation
  - Fluid, Thermal, Fluid-structure and Acoustic solvers
  - 3D design and optimization

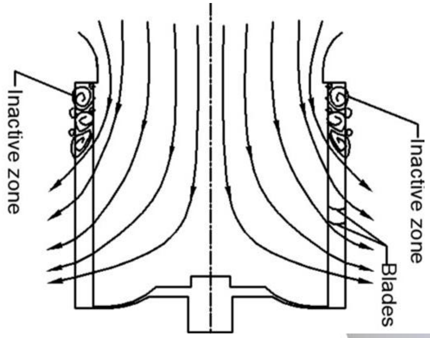
cadence®

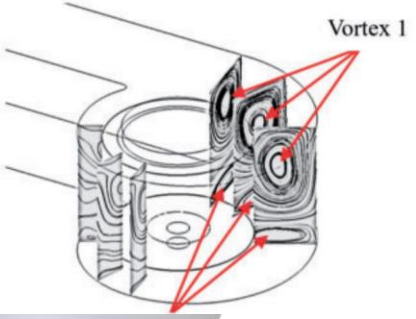
fan2022
cadence

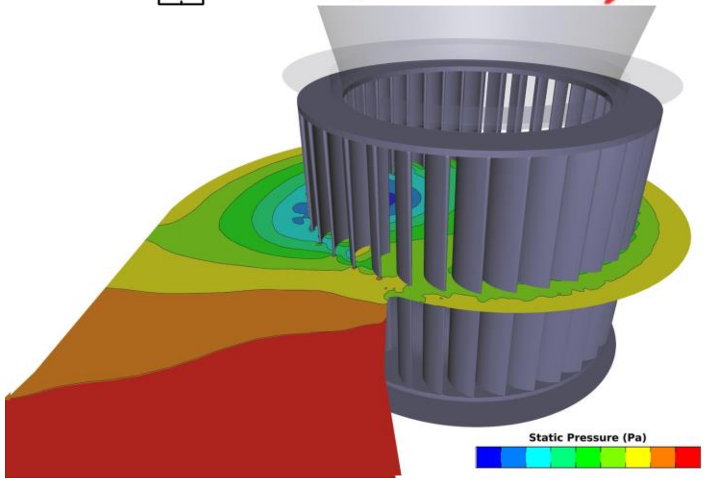
Squirrel Cage Fan

- **Squirrel Cage fans** are forward-curved centrifugal fans and commonly used in heating, ventilation and air-conditioning applications.
- They are characterized by their small size and low noise levels.
- The flow inside a squirrel cage fan is found to contain **flow separations and complex flow structures**.









Squirrel Cage Fan – Problem Definition

Problem

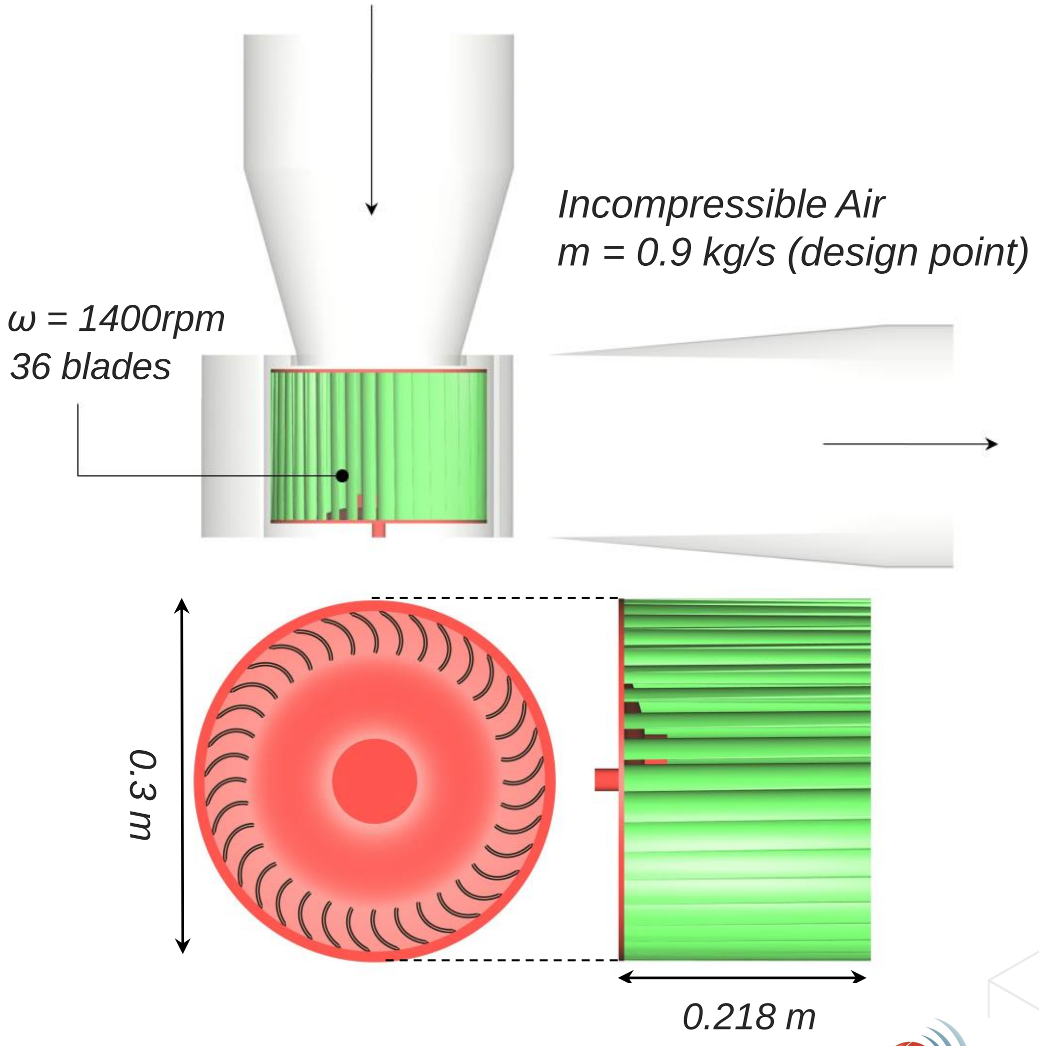
Analyzing the complex flow features in a squirrel cage fan (impeller + volute) through CFD analysis

Steps

- The geometry and computational domain are created with the information available in *Wang, 2019*
- Generated the mesh and ran a simulation using Cadence® Fidelity™ CFD
- Post-processed the flow in the squirrel fan
- Compared the data available in literature and the one achieved with Fidelity

Modelling Challenges

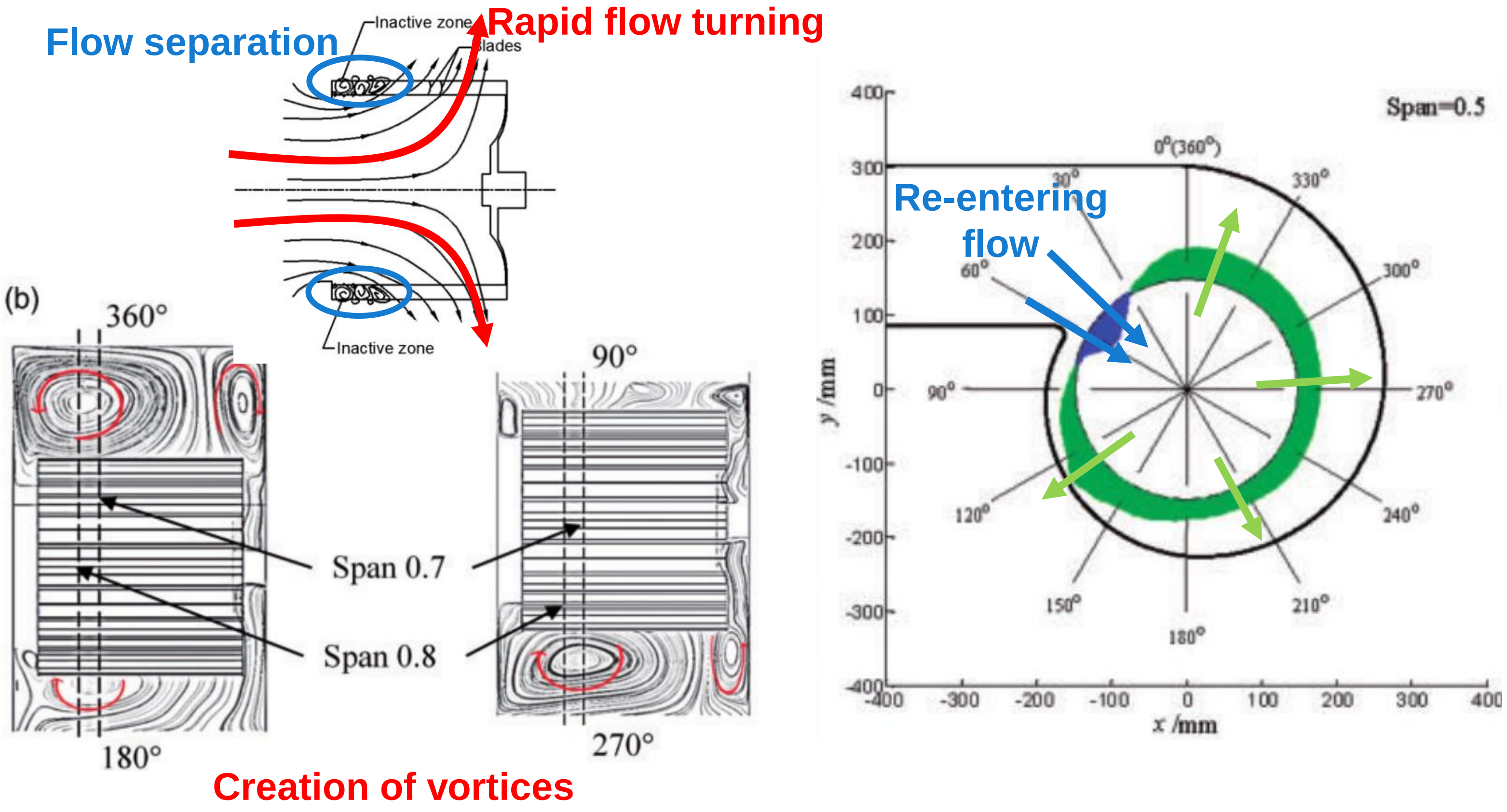
- Strong impeller-volute interaction
- Highly recirculating and complex flow inside impeller



Complex flow inside a squirrel cage fan

The flow field in squirrel cage fans is characterized by:

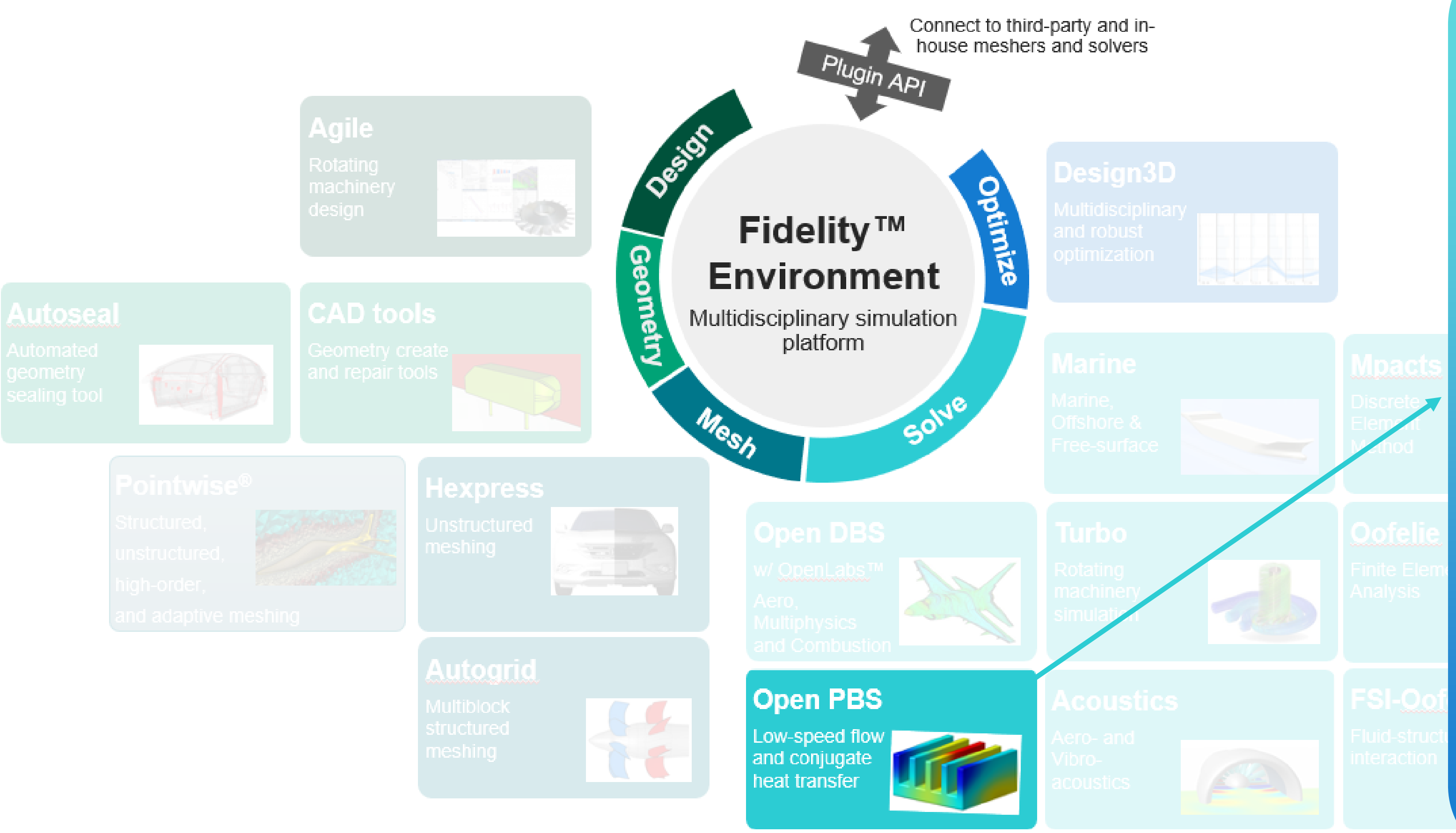
- An **inactive zone of flow separation** caused by the high curvature of the flow paths and the brusque change of direction, from axial to the radial
- **Strong interaction between the impeller and the volute**, especially near the volute tongue



fan2022
cadence

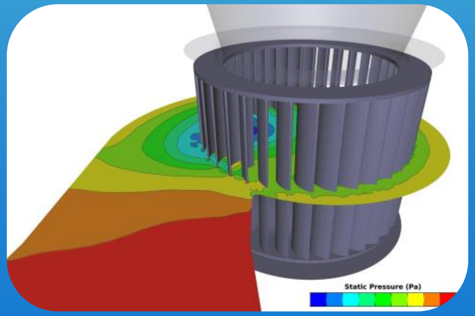
Introducing the Cadence® Fidelity™ Platform

The CFD analysis will be performed in Fidelity. The **Fidelity platform** offers a modern, flexible and user-friendly software environment containing all CFD software in a streamlined workflow



Fidelity Flow
Pressure-based solver

- **Fully coupled** (pressure & momentum solved together)
- Implicit **all-Mach** methodology
- **Highly robust and fast** convergence for a large number of applications, including industrial fans

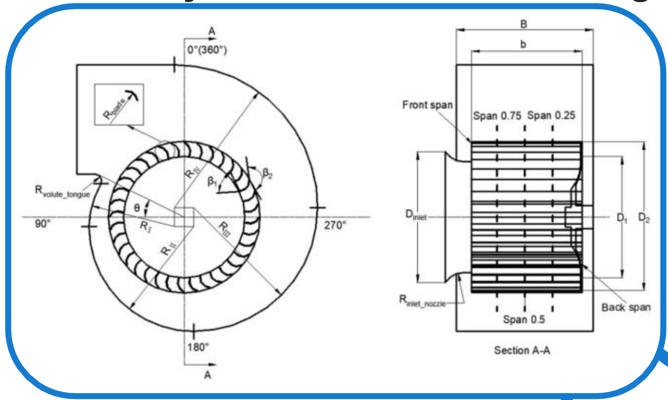


cadence

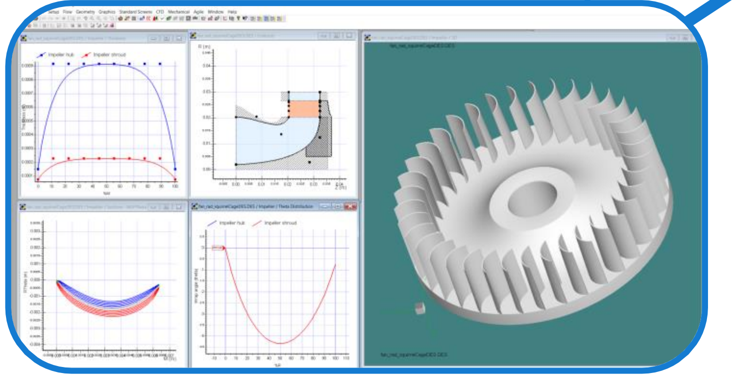
Squirrel cage fan in Fidelity

A parametrized first geometry (impeller+volute) is recreated in **AxCent** (Concepts NREC) based on the dimensions available in the paper. The **CAD geometry is imported in Fidelity**, and **two fluid domains** are generated and connected for running the CFD analysis.

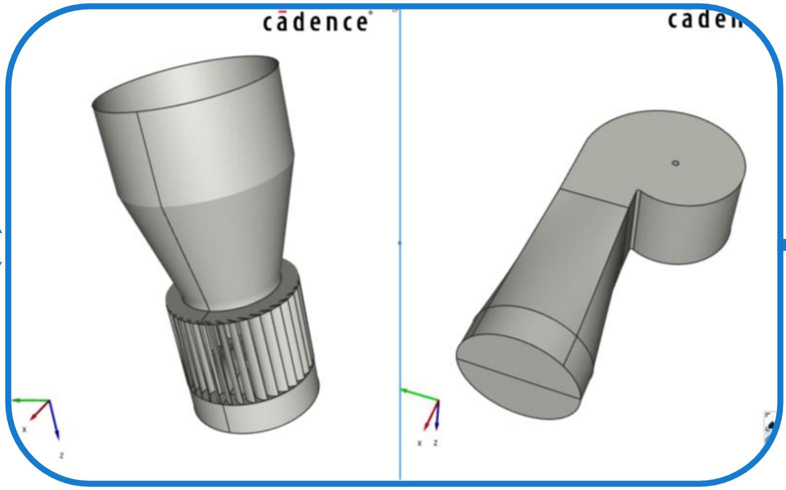
Geometry schematics from Wang



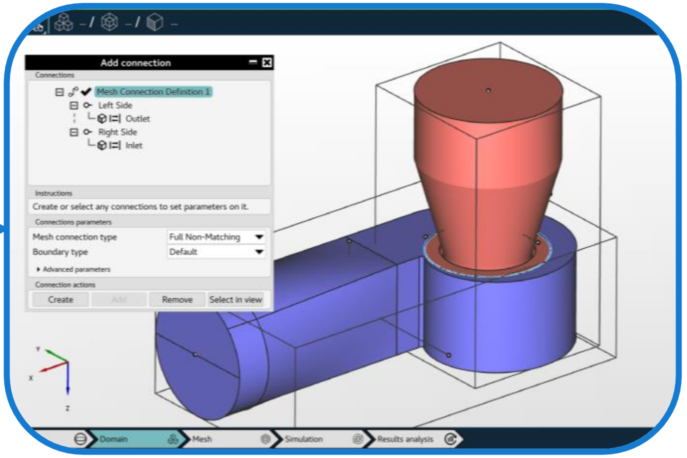
AxCent geometry reconstruction



Direct CAD import in Fidelity

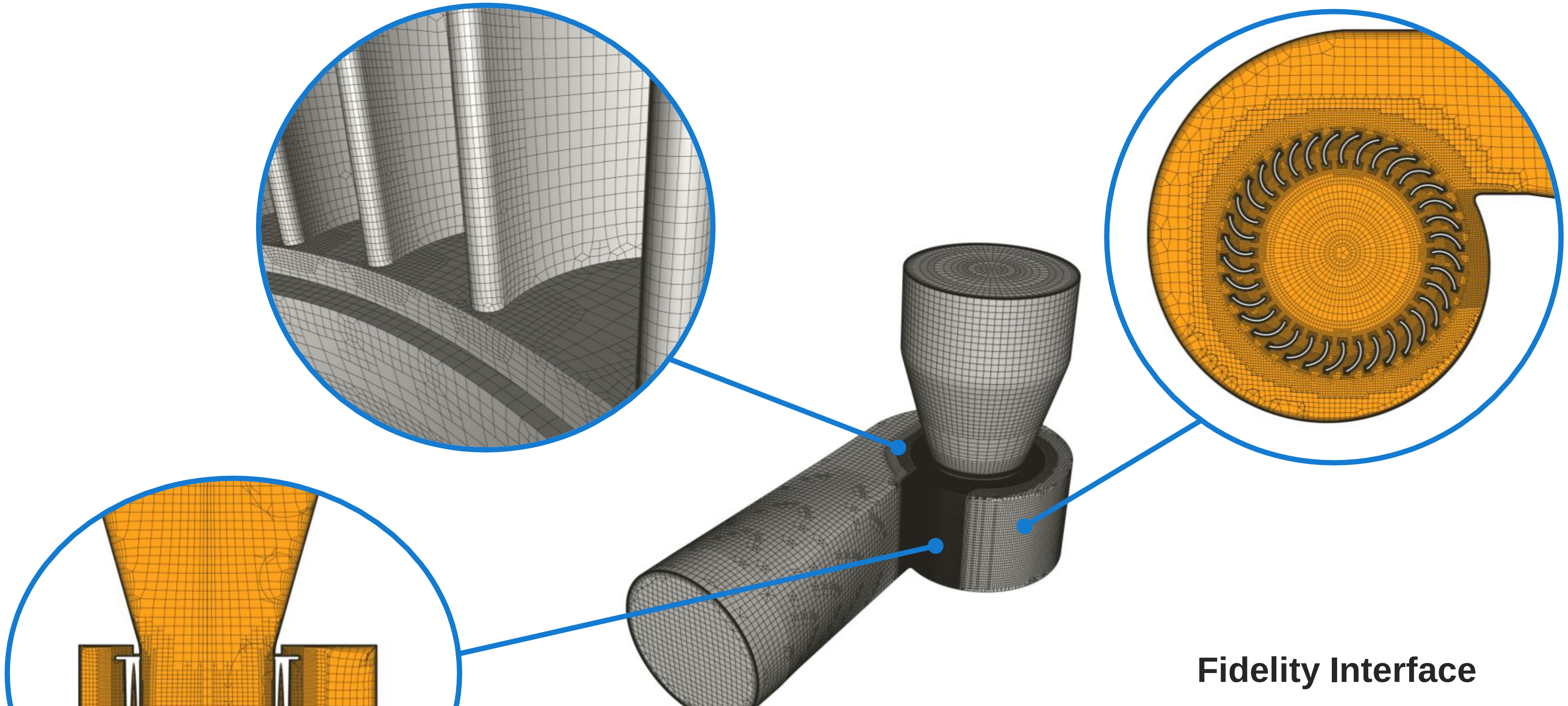


Define impeller & volute domain with a R/S connection in between



Unstructured mesh using Fidelity Automesh

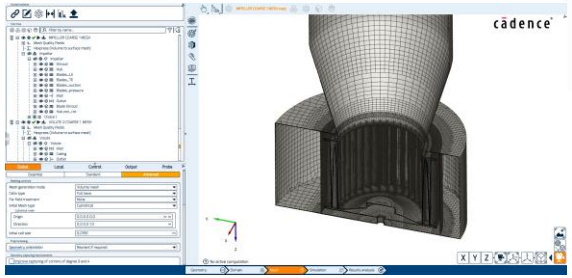
A low-Re **high-quality unstructured mesh** is generated for the impeller (360°) and volute domain of the squirrel cage fan. A volume-to-surface approach is used with hexahedra mesh cells.



Mesh quality

Mesh count	24.4M
Max expansion ratio	13
Max skewness	0.95

Fidelity Interface



Fidelity Automesh
Parallel Unstructured
Hex Dominant Meshing
for complex and/or
unclean geometries

Fidelity Flow – PBS
Multiphysics multipurpose CFD solver
dedicated to complex internal
and external flows

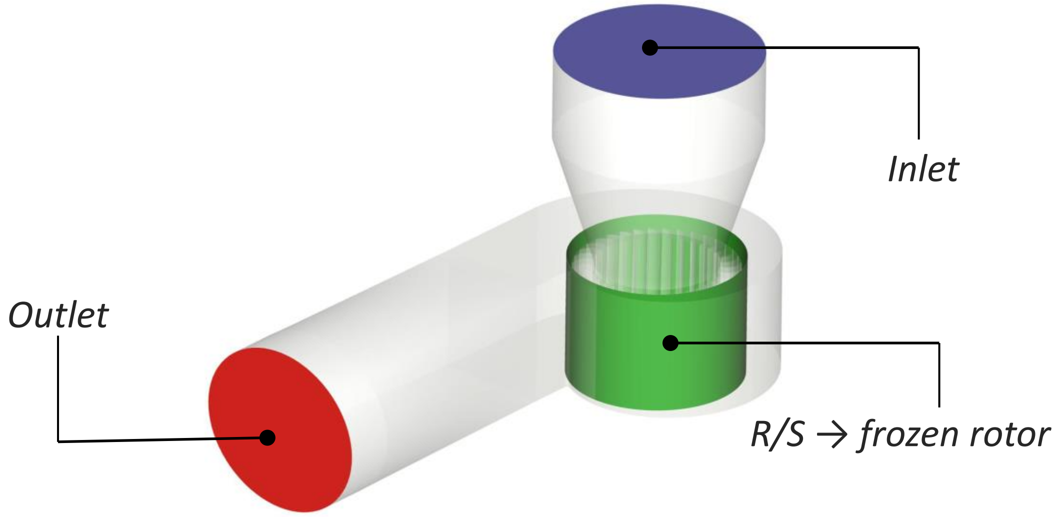
Fidelity Post-pro
Polyvalent parallelized Computational Flow
Visualization software system with turbomachinery,
acoustics, multiphysics specific features

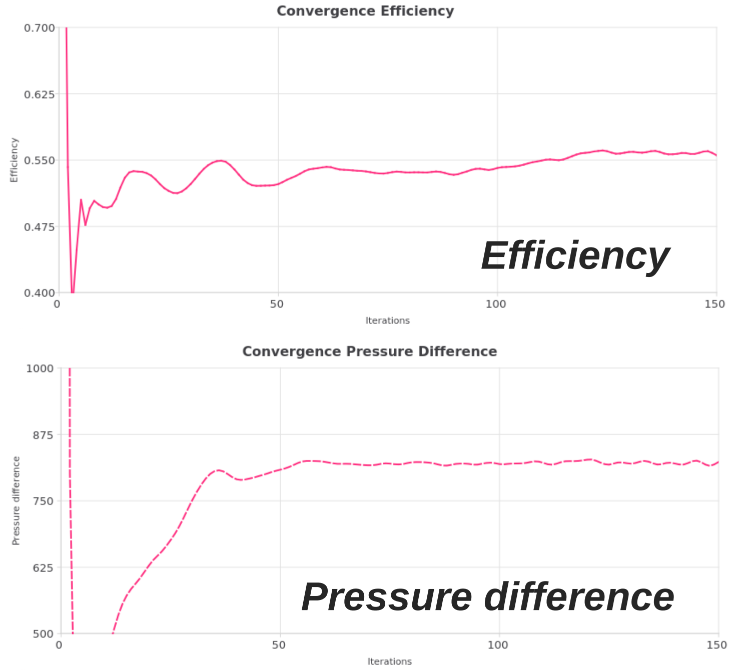
fan2022
cadence

Simulation Setup in Fidelity flow - PBS

The pressure-based solver in Fidelity is selected due to its **high performance** and **accuracy** in case of low-speed and incompressible flows.

General settings	**Turbulence model**	K-ω SST (Extended wall functions)
	Fluid	Air (incompressible)
	Inlet BC	Mass flow imposed (0.9kg/s), normal to the inlet
	Walls	Fan - Rotation speed and adiabatic (1400rpm) Volute - Static and adiabatic
	Outlet BC	Static pressure imposed (1atm)
	Time configuration	Steady
	R/S	Frozen rotor





Fidelity Automesh

Parallel Unstructured Hex Dominant Meshing for complex and/or unclean geometries

Fidelity Flow – PBS

Multiphysics multipurpose CFD solver dedicated to complex internal and external flows

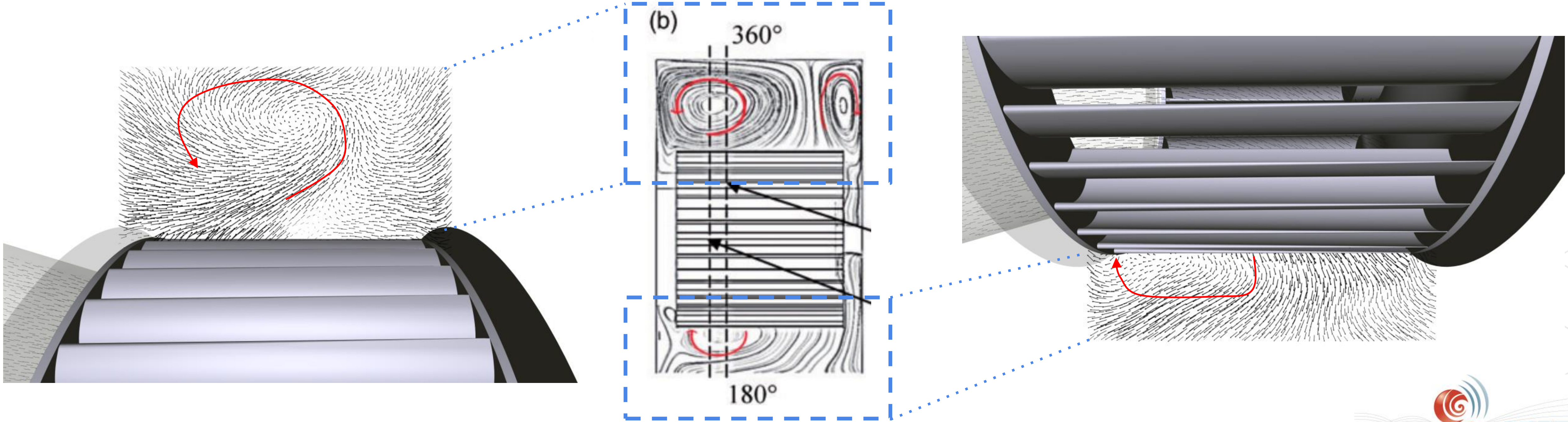
Fidelity Post-pro

Polyvalent parallelized Computational Flow Visualization software system with turbomachinery, acoustics, multiphysics specific features

fan2022 cadence

Flow Analysis

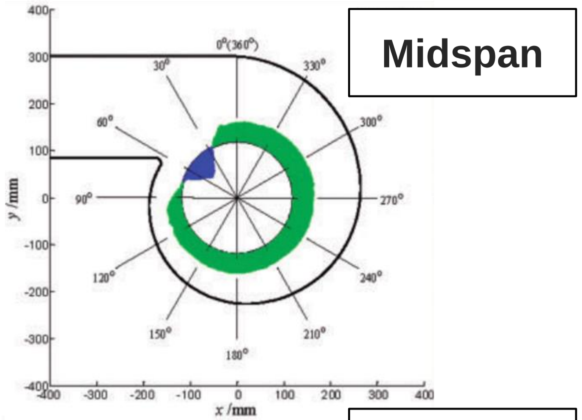
- The flow field in squirrel cage fans is characterized by **an inactive zone of flow separation** near the impeller tip due to rapid flow turning from axial to centrifugal
- The recirculating flow pattern is also found in the volute during the CFD analysis. The rapid flow turning comes with a negative radial velocity at the tip resulting in the creation of a **vortex** between 50%-100% span height in the volute.
- *Wang* also shows the presence of a **2nd vortex** growing towards the volute outlet (360°) due to the clearance between the hub and the volute



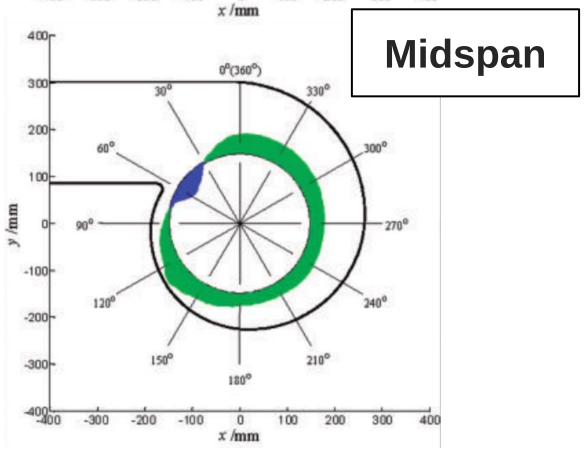
Flow Analysis

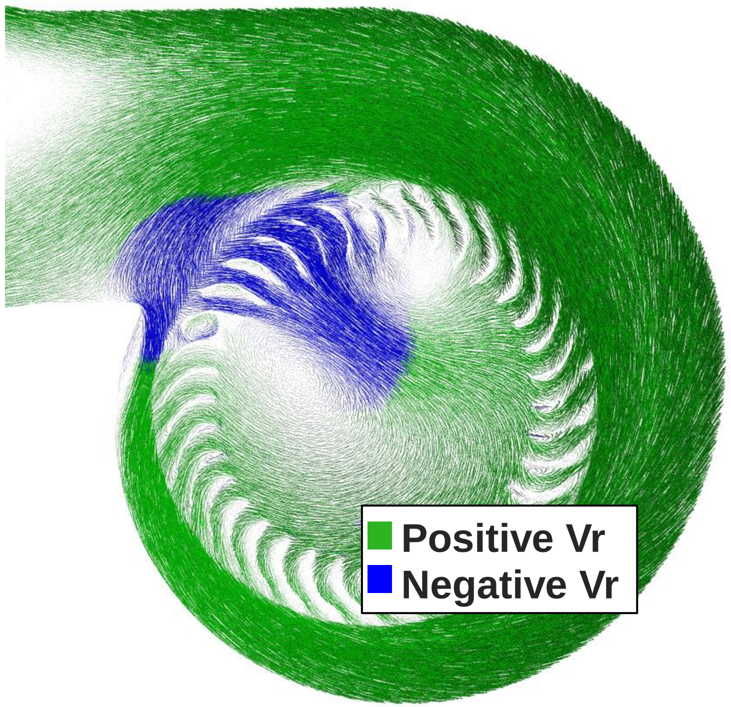
- The flow field in squirrel cage fans is characterized by a **strong interaction between the impeller and the volute**, especially near the volute tongue
- At midspan near the tongue, the flow field in the impeller shows a **negative radial velocity** over a short tangential distance.
- The same flow pattern is found in the CFD results. As for the paper, the tangential distance with negative V_r is higher at the impeller outlet, but the CFD simulation overpredicts the tangential extent of the negative radial velocity

Vr
Impeller
Inlet

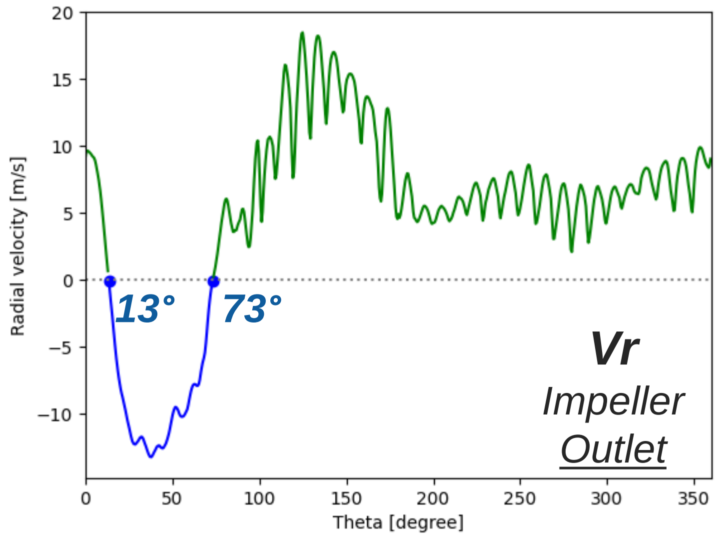


Vr
Impeller
Outlet





Velocity vectors colored by Vr



fan2022
cadence

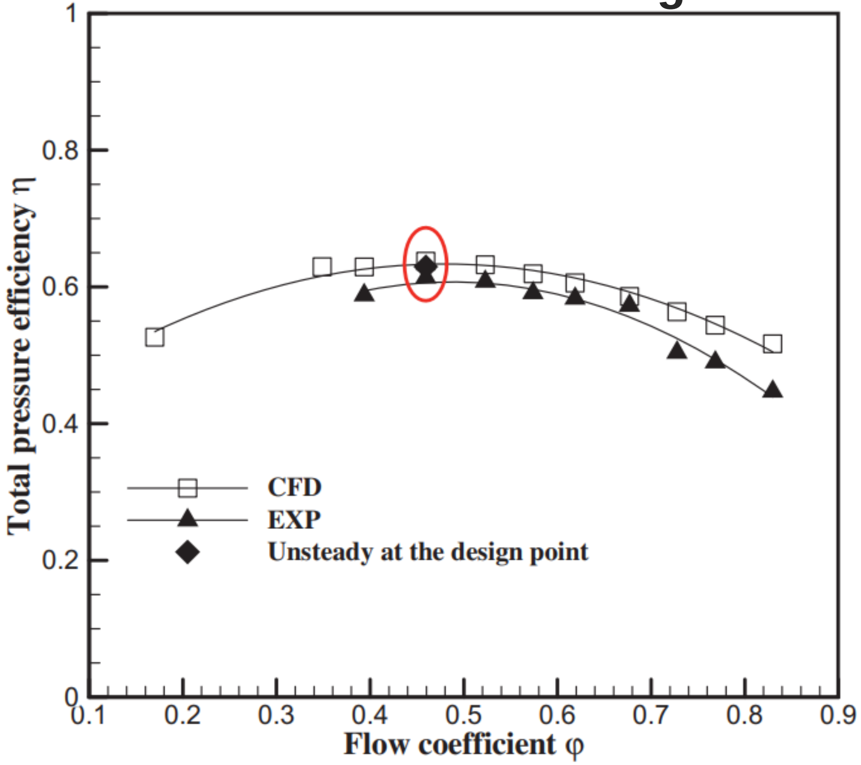
Flow Analysis

- A **complete speedline** can be created by varying the mass flow at the inlet. The mass flow is varied between 0.3 and 1.8 times the design mass flow and shown as the flow coefficient in function of the total pressure efficiency

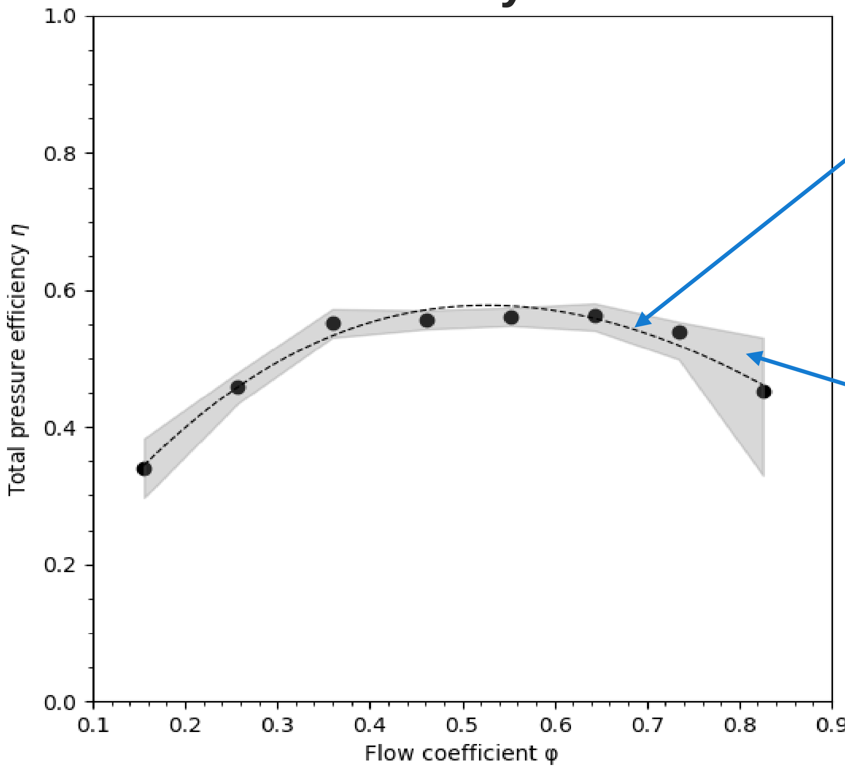
$$\varphi = 4Q/\pi D_2^2 u_2 \qquad \eta = \Delta P_t Q/W_{in}$$

- A **good agreement** can be found with the shape of the paper speed line. The small differences in absolute values are due to small differences in geometry and the uncertainty concerning the ΔP_t measuring points

Data from Wang



Fidelity

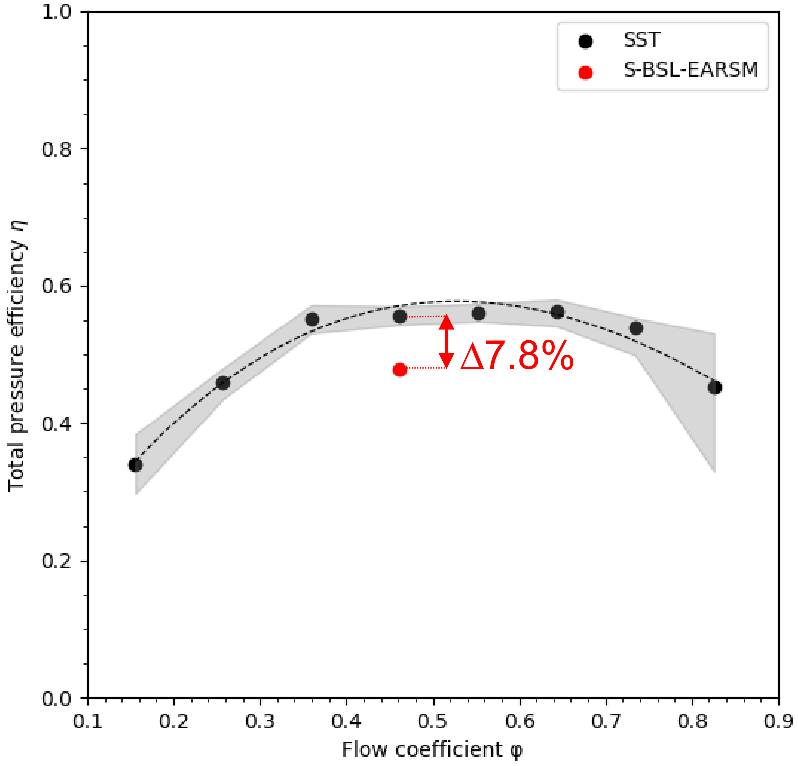


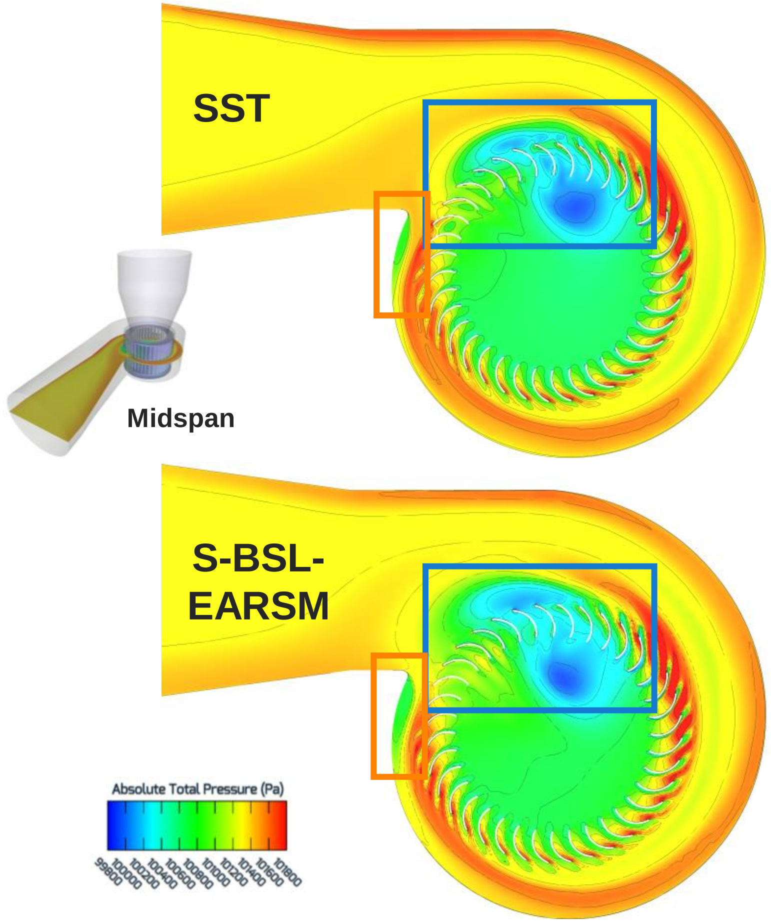
Speedline at 1400rpm

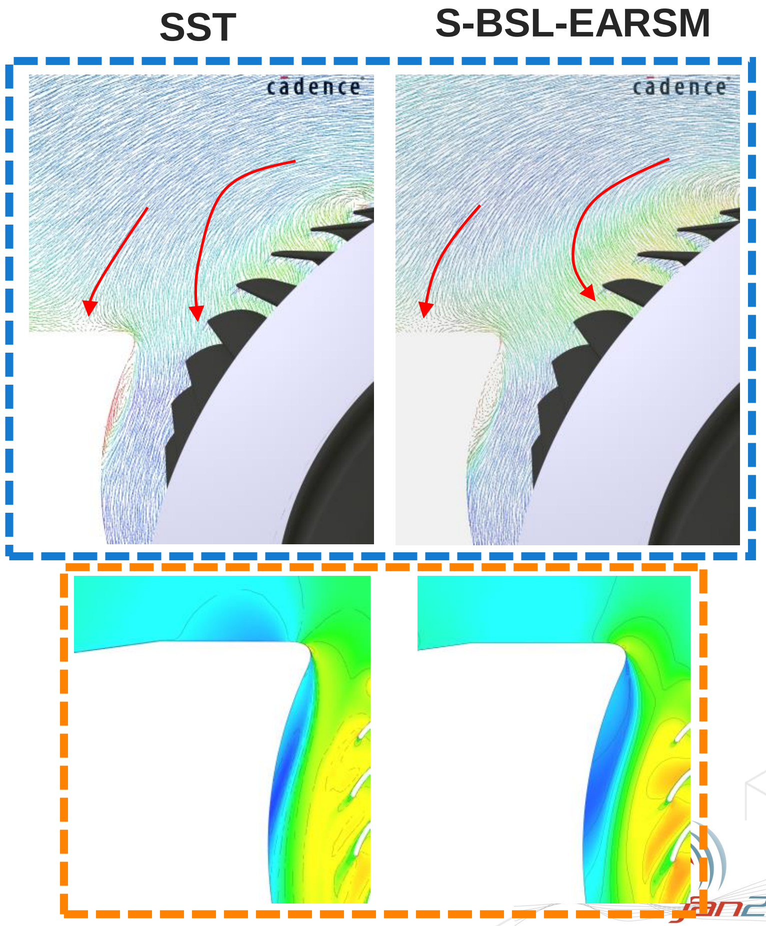
Grey backgound represents the oscillating behavior of the steady simulation convergence (due to the unsteady nature of the flow)

Flow Analysis – Turbulence model comparison

- The design point has been evaluated with a different non-linear eddy viscosity 2-equation turbulence model: **S-BSL-EARSM**
- The high efficiency difference compared to SST can be explained by the **higher losses in the volute**. A similar pressure rise in the impeller is noted, but a different flow field at the volute tongue and in the region of the re-entering flow leads to a higher dissipation of the generated head, lowering the overall efficiency.



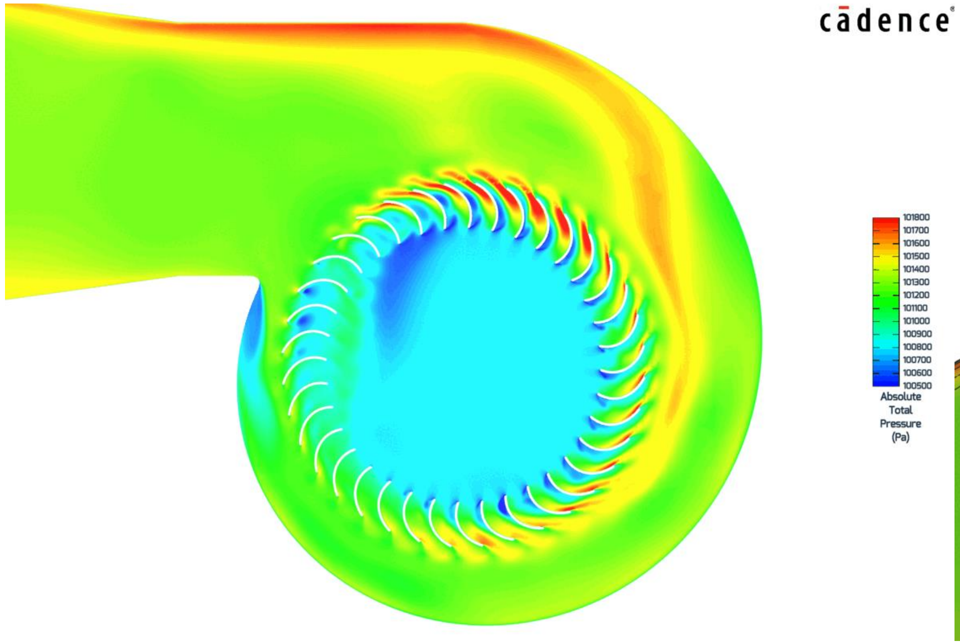


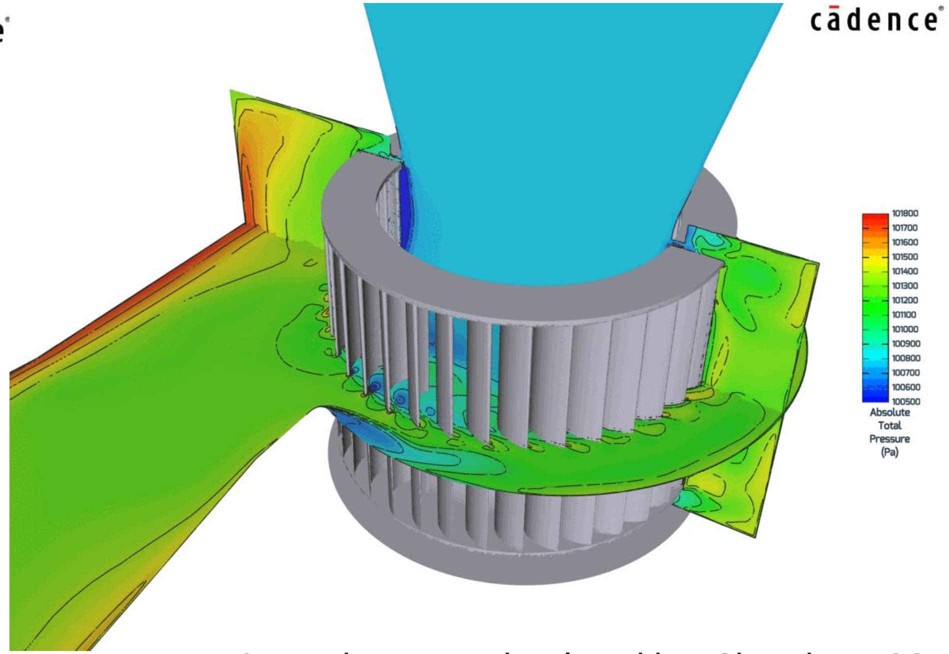


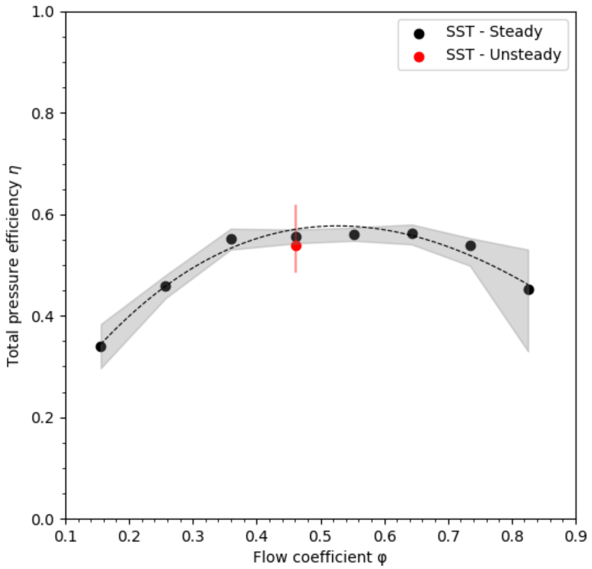
cadence

Flow Analysis – Unsteady simulation

- Due to the unsteady nature of the flow inside the squirrel cage fan, the design point is evaluated with an **unsteady simulation**
- In order to speed up the convergence of the unsteady simulation in Fidelity, **the number of steps per revolution is modified dynamically** during the simulation. A final time step of 1° is considered.
- The global performances of the unsteady simulation are similar to the ones of the steady simulation







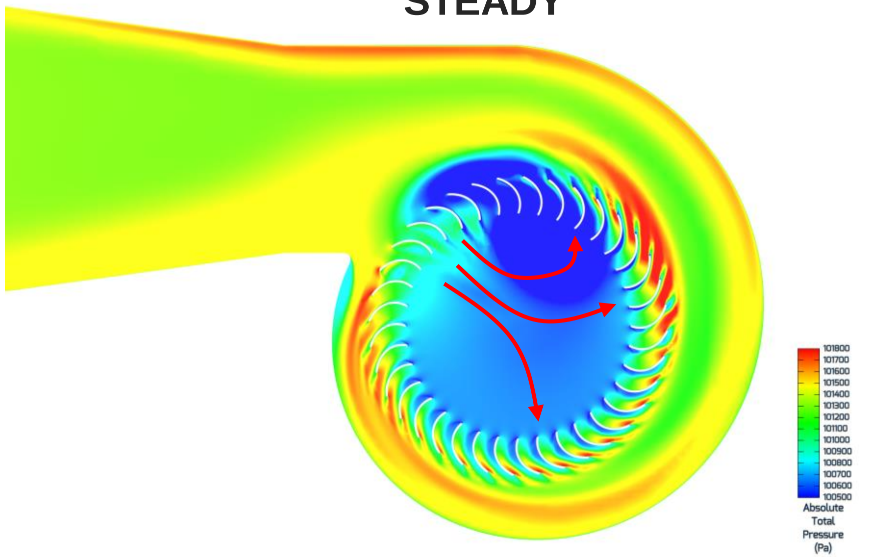
10 rotations are simulated in 12h using 192 procs:

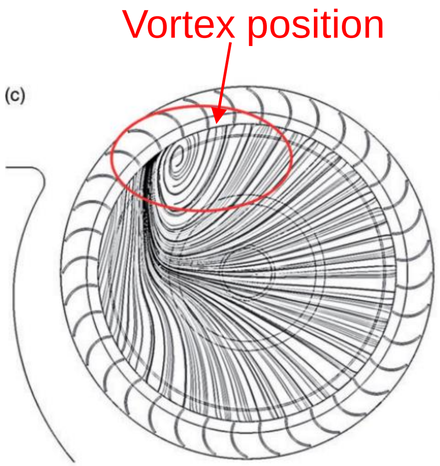
- 2 rotations with 4° angular resolution
- 2 rotations with 2° angular resolution
- 6 rotations with 1° angular resolution

Flow Analysis – Unsteady simulation

- Due to the unsteady nature of the flow inside the squirrel cage fan, the design point is evaluated with an **unsteady simulation**
- In order to speed up the convergence of the unsteady simulation in Fidelity, **the number of steps per revolution is modified dynamically** during the simulation. A final time step of 1° is considered.
- The flow field between the steady and unsteady simulation is slightly different at the location of the impeller-volute interaction near the volute tongue. **The location of the vortex and the velocity field** as shown by the paper, are better predicted by the unsteady simulation. Also, the tangential extent of the negative radial velocity now better matches the paper results.

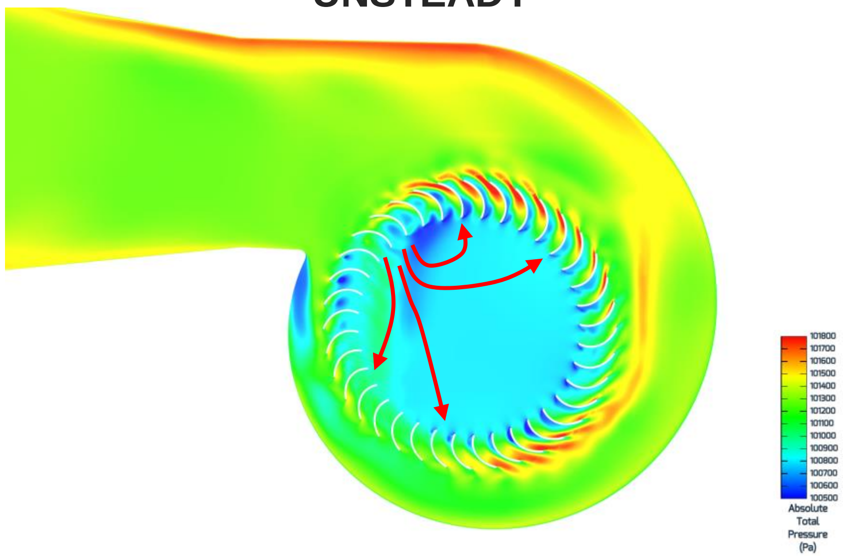
STEADY





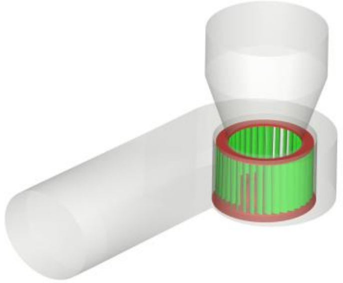
Streamlines at midspan (Wang)

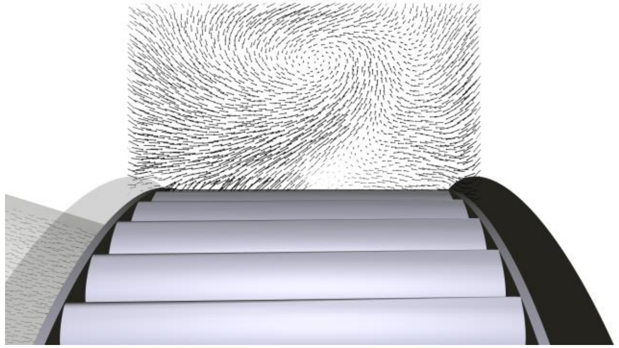
UNSTEADY

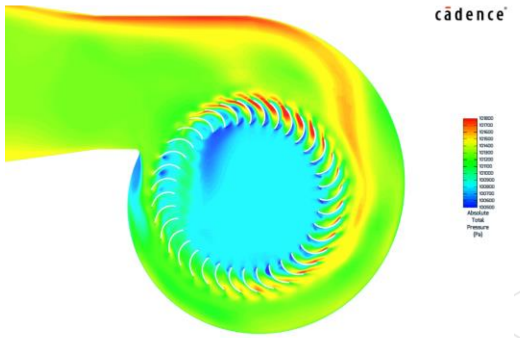


Conclusions

- **The flow inside a squirrel cage fan is highly complex** and characterized by recirculating flow and strong component interaction
- A squirrel cage geometry found in literature was rebuilt, meshed and analyzed through CFD simulations in the Cadence design systems platform **Fidelity**. The results are compared to experimental and numerical data from a paper
- A fast **coupled pressure-based solver** is selected in Fidelity to run the CFD simulations due to its high performance and accuracy in case of low-speed and incompressible flows.
- **Steady CFD simulations** of design point and the speed line show a **good agreement** in flow structure and global performances compared to the paper results.
- The design point is evaluated with 2 different turbulence models: the standard **SST turbulence model and S-BSL-EARSM**. Both models lead to a similar flow field in the impeller, but S-BSL-EARSM is predicting higher losses in the volute
- Finally, the unsteady nature of the flow is modelled in more detail through an **unsteady simulation at design point**. A **variable time step** allows the designer to speed up the simulation convergence. The unsteady simulation shows similar global performances as the steady simulation, but the flow field is better matching the paper results







fan2022
cadence

Q&A

References

- *Wang, Ke, Yaping Ju, and Chuhua Zhang. "Numerical investigation on flow mechanisms of a squirrel cage fan." Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 233.1 (2019): 3-16.*
- *Mangani, Darwish, Moukalled. Development of a pressure-based coupled CFD solver for turbulent and compressible flows in turbomachinery applications." Proceedings of ASME Turbo Expo 2014, GT2014-25967*

cādence®