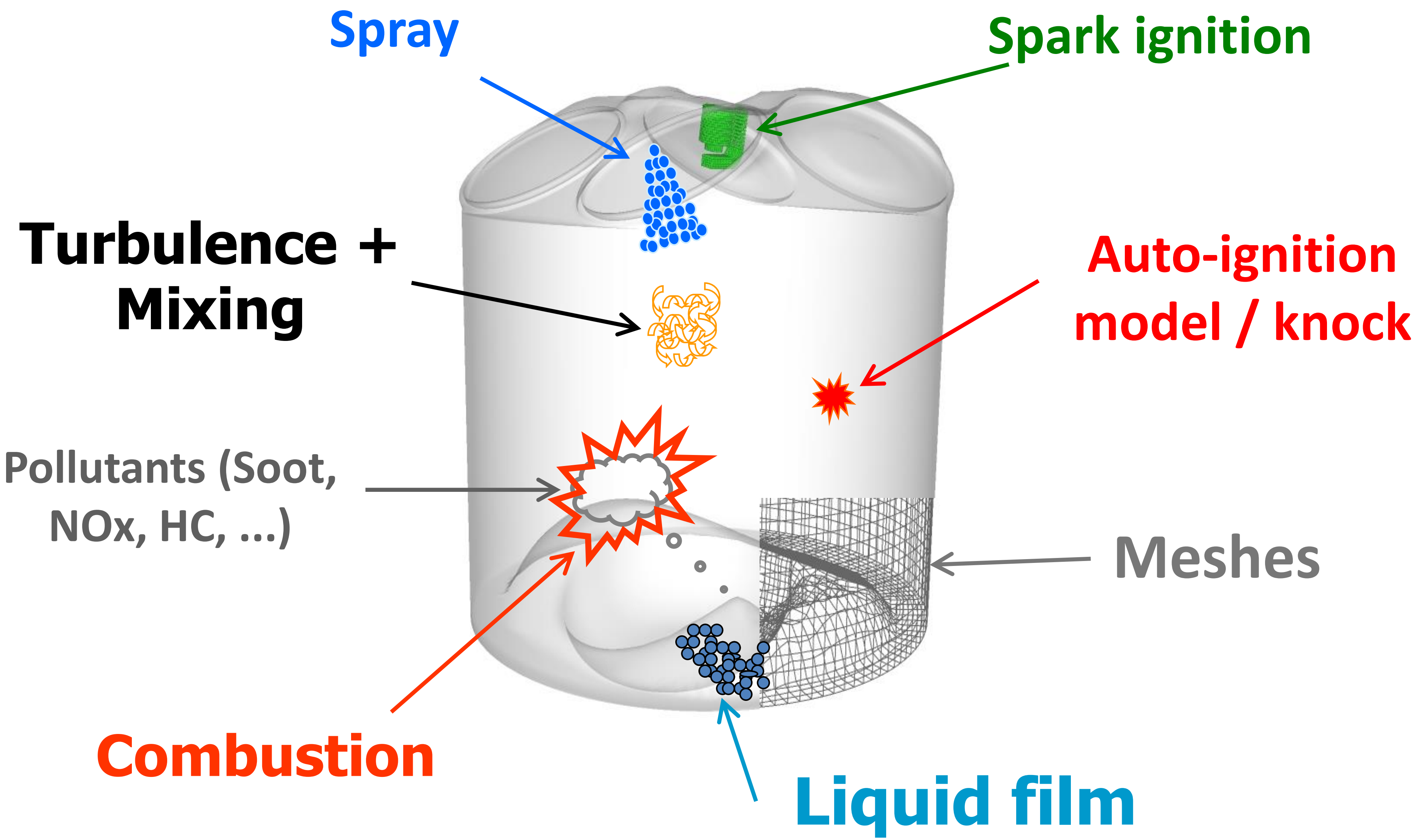
EFC Topic 4.2
Simulation-to-Simulation Benchmarking (LES, RANS)

Presented by Cecile PERA (IFPEN)

Objectives of the 4.2

Summarize: some guidelines for engine simulations

Introduction



Contributions

1. Meshing strategy

Cecile Pera, cecile.pera@ifpen.fr	**AVBP, IFP-C3D**
Tommaso Lucchini, tommaso.lucchini@polimi.it	**Lib-ICE**
Andreas Kempf, andreas.kempf@uni-due.de	**OpenFOAM, PsiPhi**
Kelly Senecal, senecal@convergecfd.com	**Converge**

2. Boundary conditions and methodology

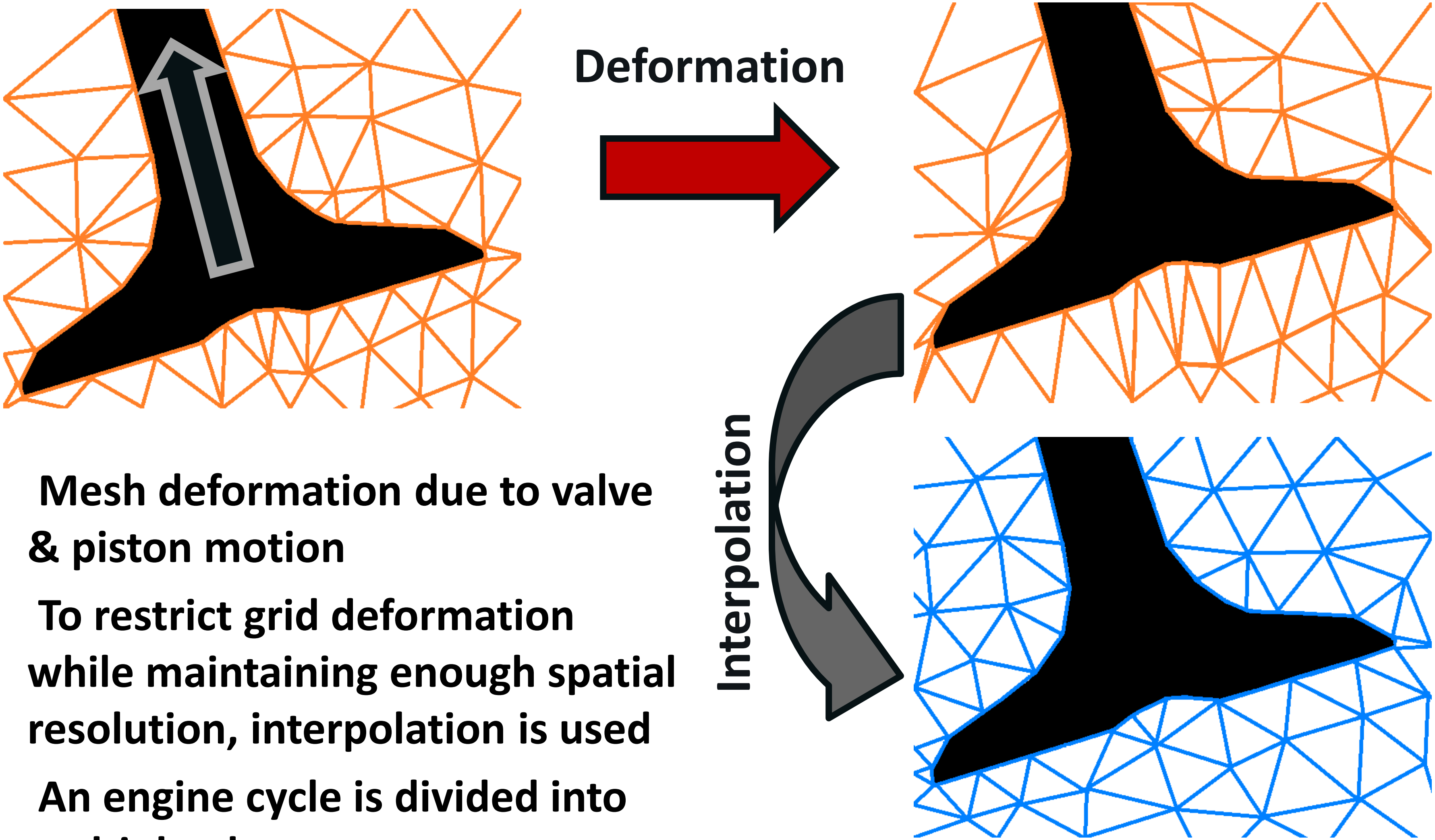
Cecile Pera, cecile.pera@ifpen.fr	**AVBP, IFP-C3D**
Tommaso Lucchini, tommaso.lucchini@polimi.it	**Lib-ICE**
Andreas Kempf, andreas.kempf@uni-due.de	**OpenFOAM, PsiPhi**
Stefano Fontanesi, stefano.fontanesi@unimore.it	**star-cd**

3. Modeling issues

Cecile Pera, cecile.pera@ifpen.fr	**AVBP, IFP-C3D**
Andreas Kempf, andreas.kempf@uni-due.de	**OpenFOAM, PsiPhi**
Kelly Senecal, senecal@convergecfd.com	**Converge**
Satbir Singh, satbirs@andrew.cmu.edu	**CMU**

1. Meshing strategy

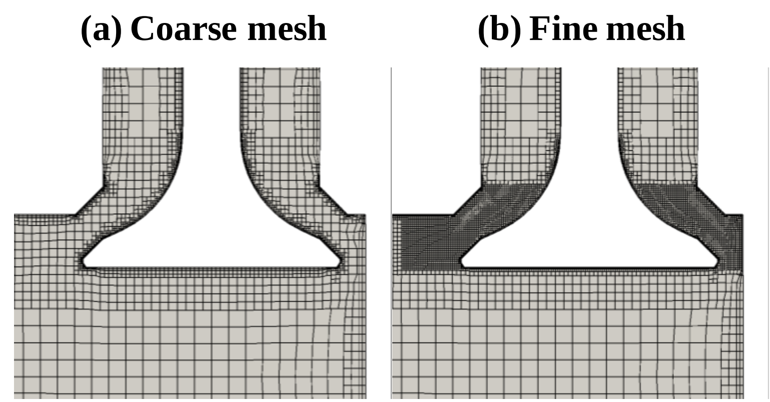
Meshing Issues: Body Conformal

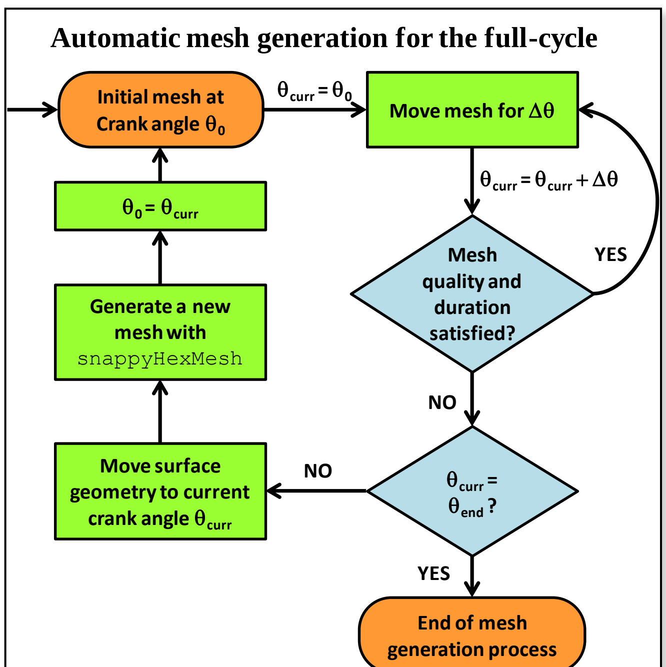


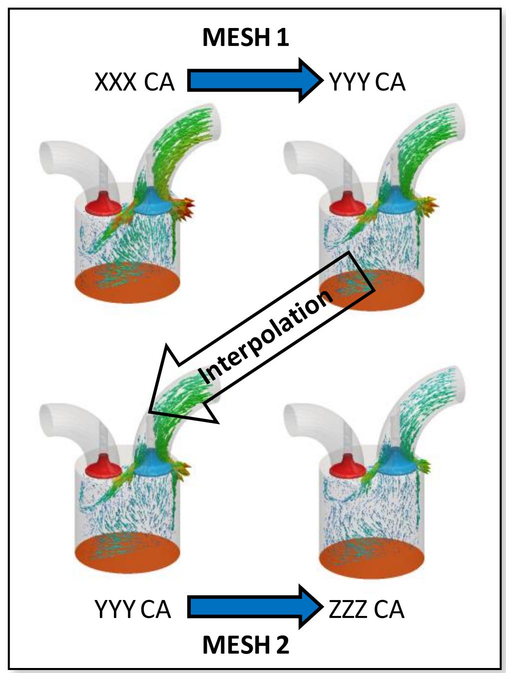
- Mesh deformation due to valve & piston motion
- To restrict grid deformation while maintaining enough spatial resolution, interpolation is used
- An engine cycle is divided into multiple phases

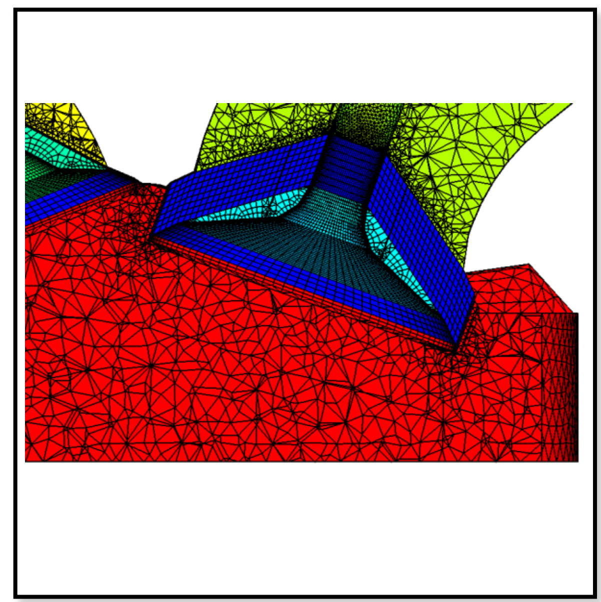
Automatic Mesh Generation

- CFD code: Lib-ICE, based on OpenFOAM technology
- CFD solver: compressible, pressure based, RANS
- Automatic mesh generation (based on snappyHexMesh) + automatic mesh motion

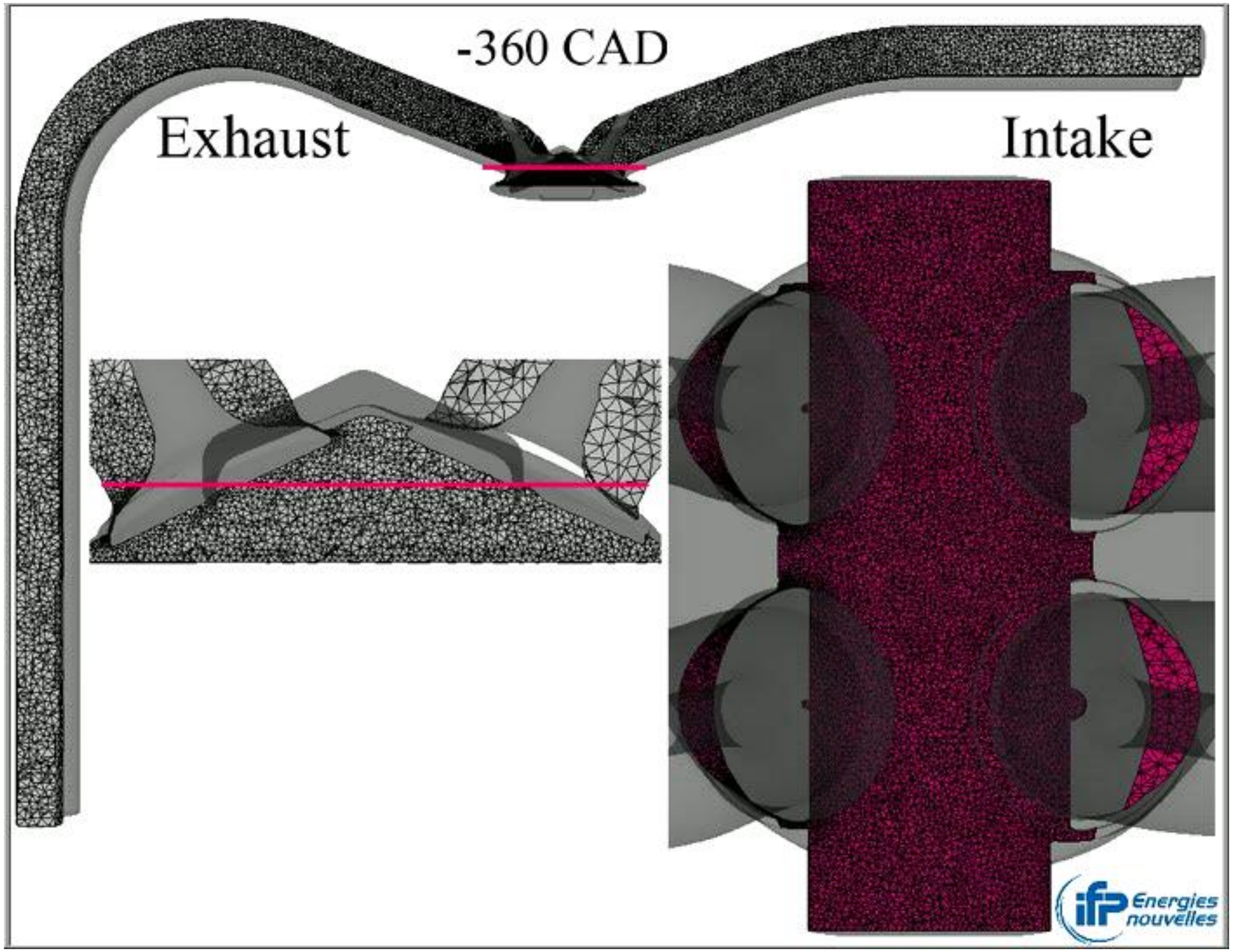






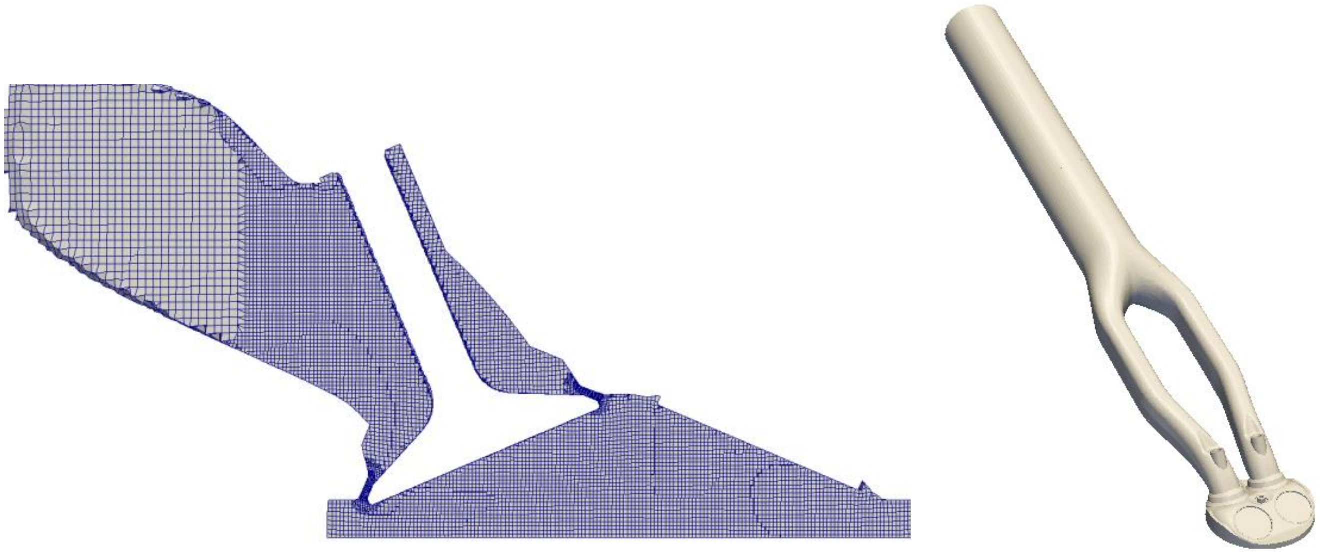


Automatic Mesh Generation



Body Conformed moving grid (OpenFOAM)

- Mesh points move to comply with piston and valve motion
- Quality of the mesh reduces during grid movement
- Local mesh-refinement reduces amount of cells

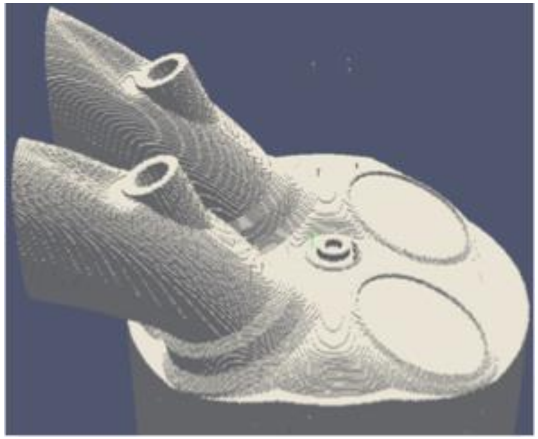


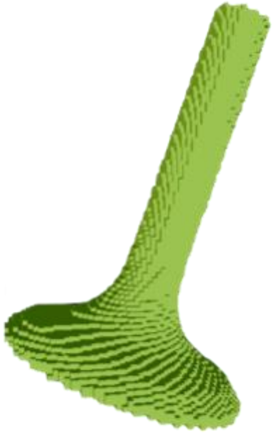
Mesh of Darmstadt Engine, intake valve plane. 0.5 mm, (P. Janas)

Immersed Boundary (PsiPhi, In-house)

UNIVERSITÄT DUISBURG ESSEN

- Immersed particles into structured Cartesian background grid
  - Cell with particle = solid
- No meshing required!
- The motion of the moving objects is governed by the background mesh
- No local grid refinement possible
  - Simplicity and efficiency for unstructured codes with less cells



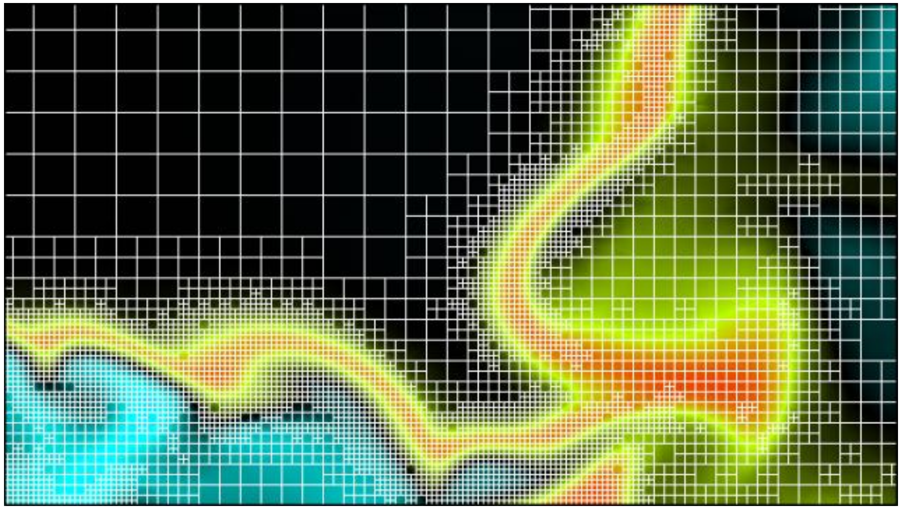


Fluid cells of the Darmstadt engine (0.3 mm) and intake valve (big voxels are shown for the valve!), (T. Nguyen)

No User Grid Generation

Import Geometry → Setup Case → Solve

- **Automated meshing**
  - **No meshing time**
- **Adaptive Mesh Refinement (AMR)**
  - **No more guessing**
- **Orthogonal cells**
- **Easy to perform grid convergence studies**



2. Boundary Conditions and Methodology

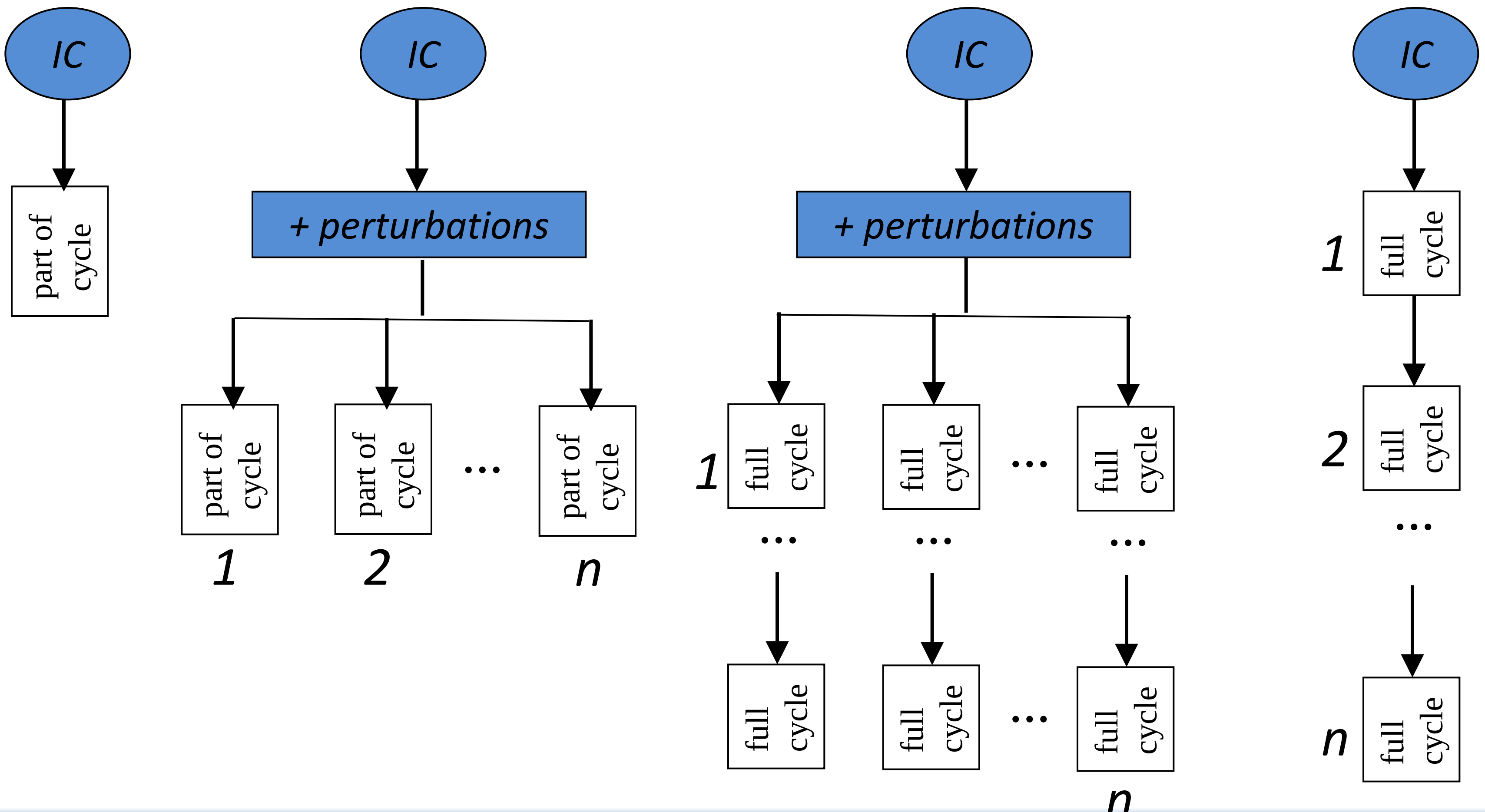
Multi-cycles LES

Single **C**ycle

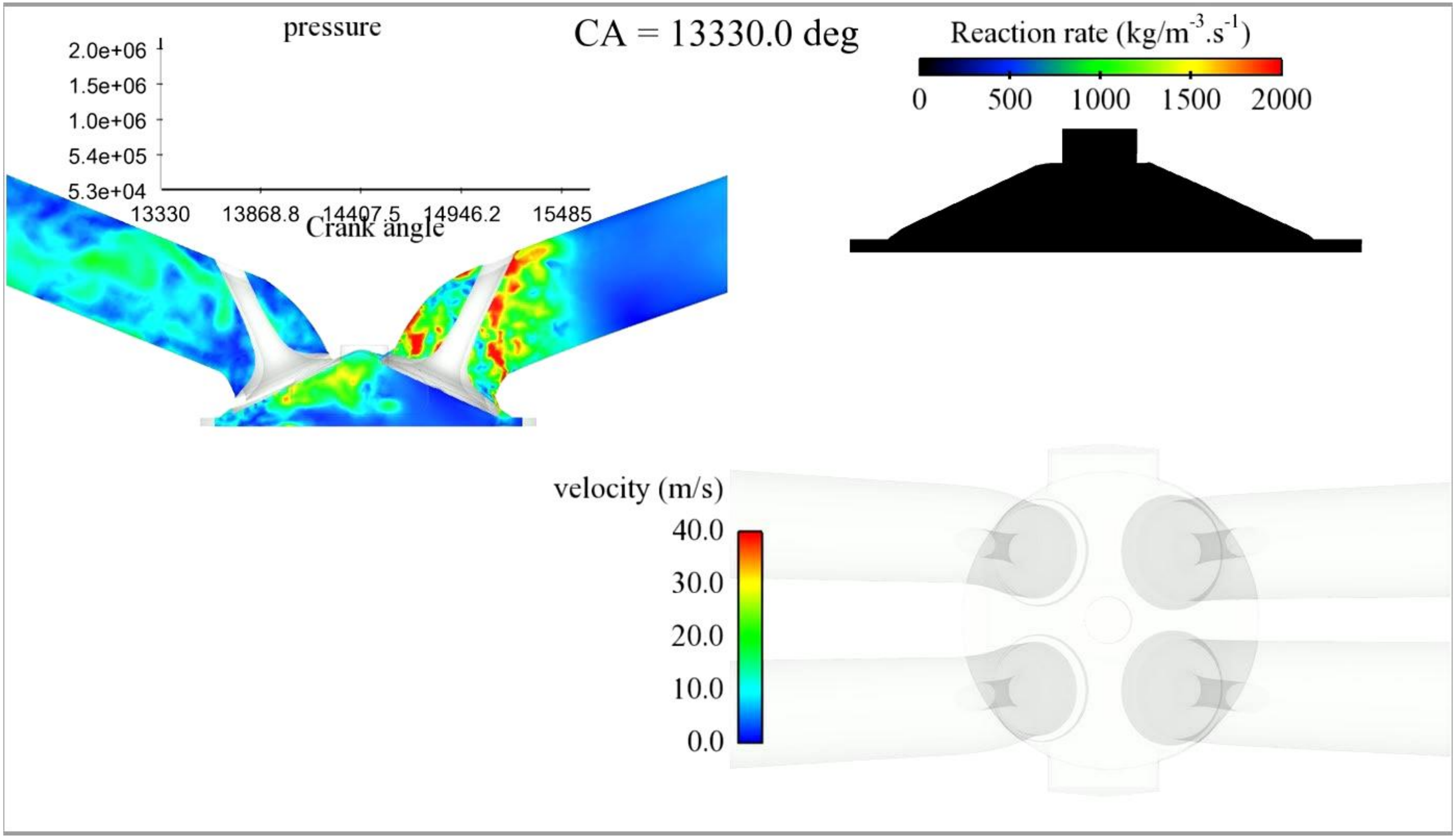
Multiple **P**erturbed **C**ycles

Multiple **P**erturbed **C**onsecutive **C**ycles

Multiple **C**onsecutive **C**ycles

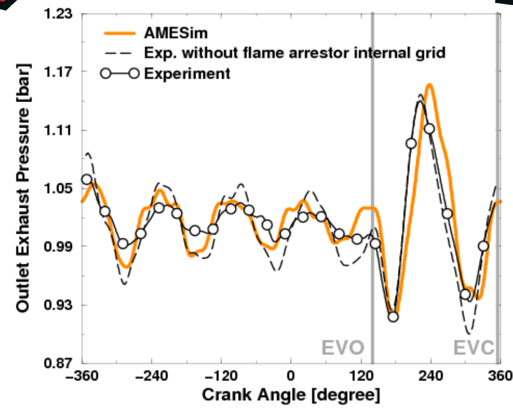


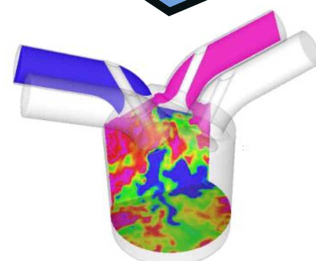
Multi-cycles LES



LES / 1D coupling

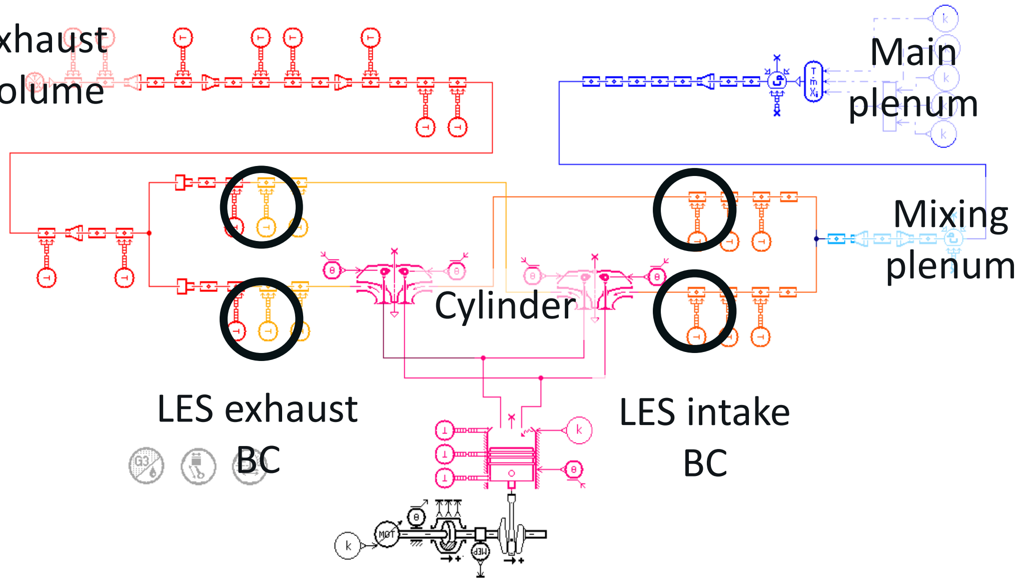
Standard engine data

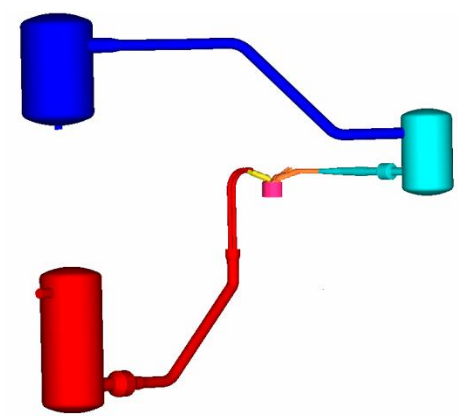




LES Engine

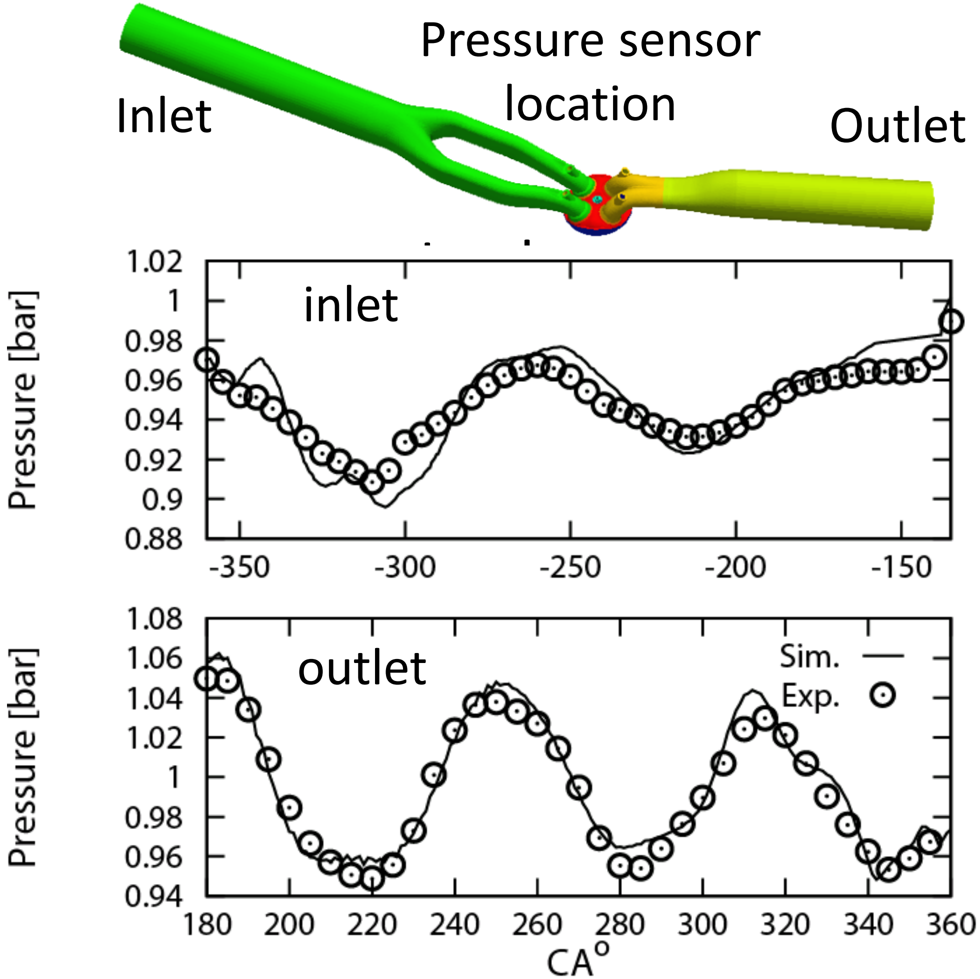
- **LES in the chamber and a part of intake and exhaust ducts**
- **Boundary condition definitions**
  - **P and T variations during engine cycle**
- **Initial states**





Navier-Stokes Characteristic Boundary Conditions (NSCBC)

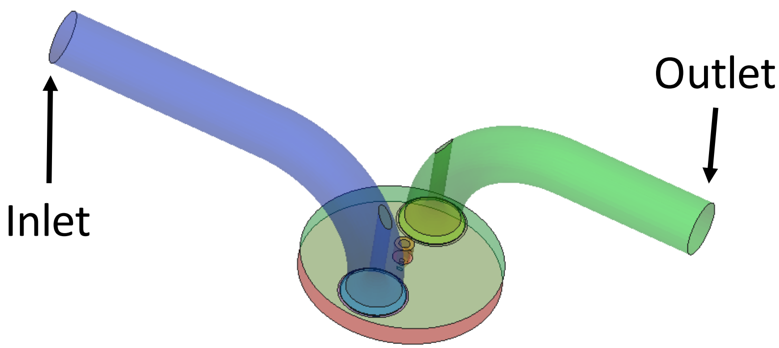
- A time-varying pressure imposed from measurements
  - Intake and exhaust ports
- Pressure from 1D acoustic simulations of manifolds
- Entire manifold system modeled (3D)
  - Simplify the boundary treatment
  - Increases computational time



In-cylinder pressure during the intake and the exhaust stroke, OpenFOAm, cold flow, 800 rpm, (P. Janas)

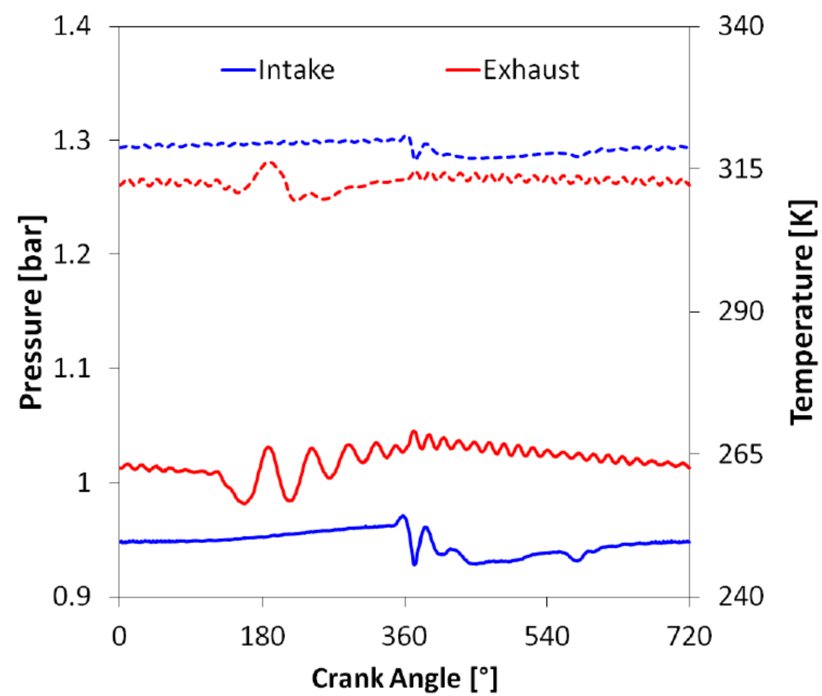
Boundary conditions (RANS TCC setup)

Simulated domain

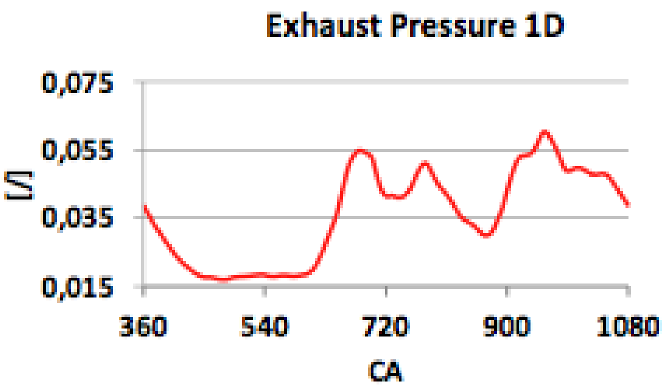


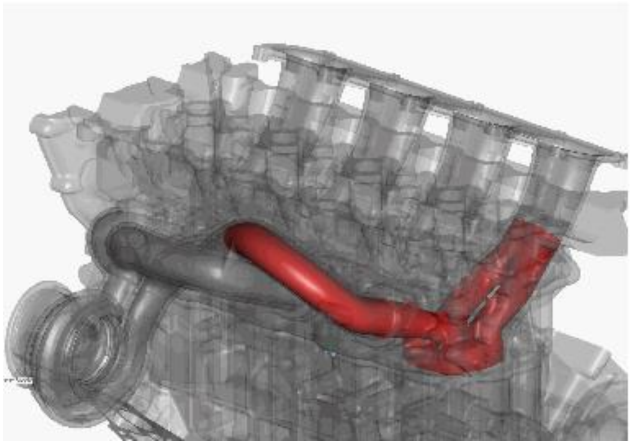
Mesh structure in the valve region

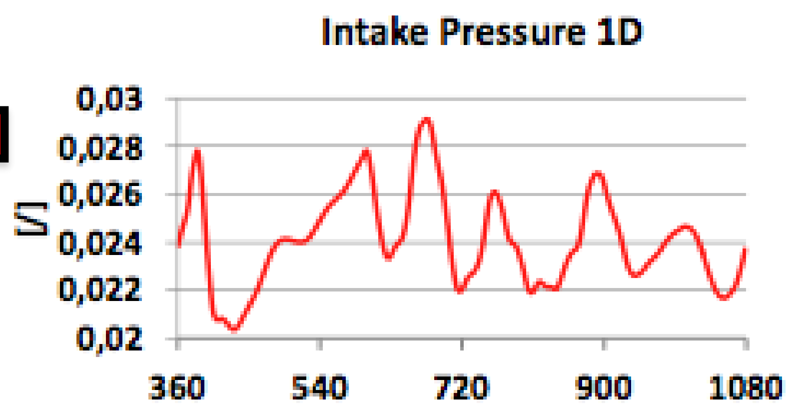
Unsteady boundary conditions at inlet and outlet boundaries



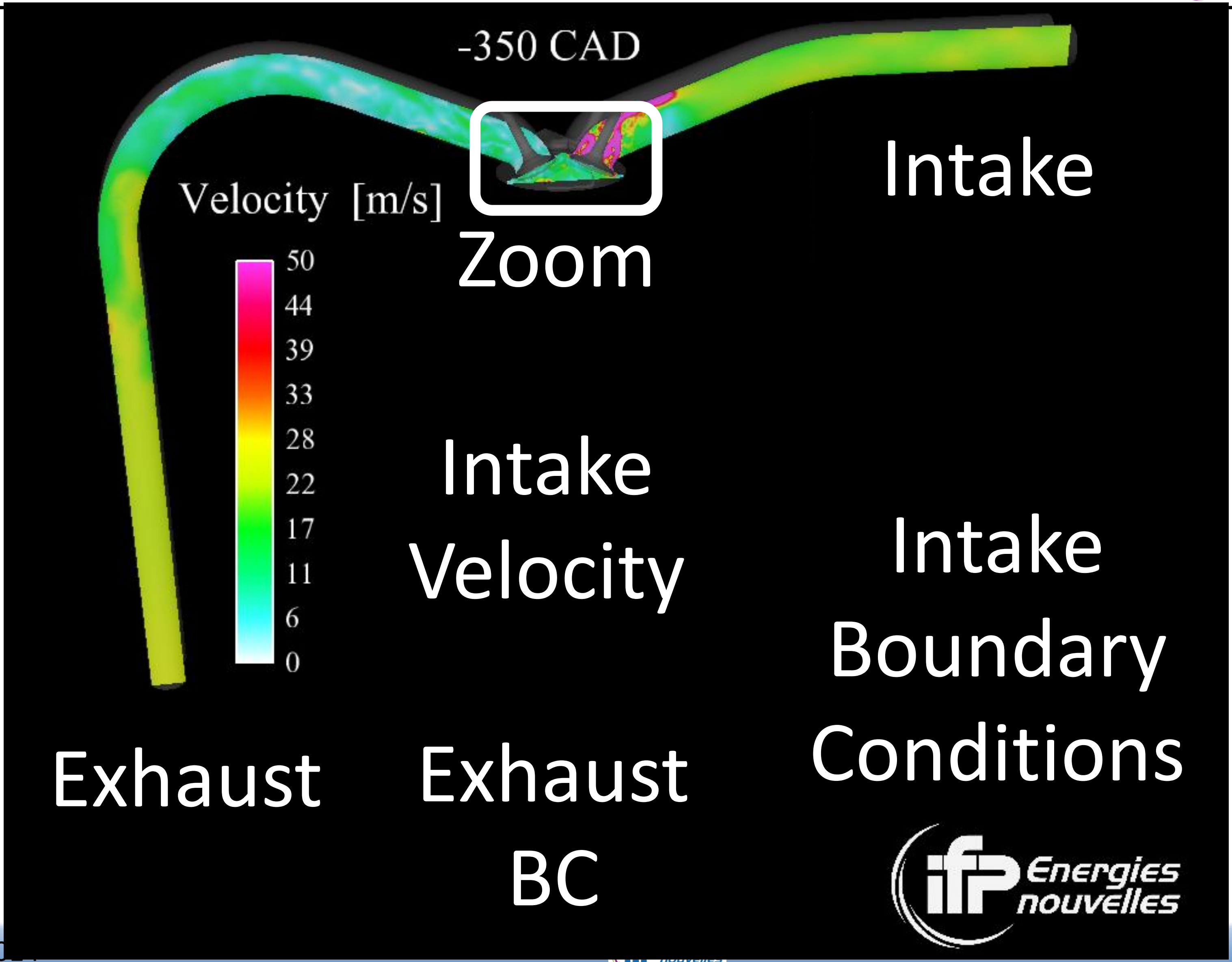
Boundary Conditions







LES / 1D coupling: acoustic

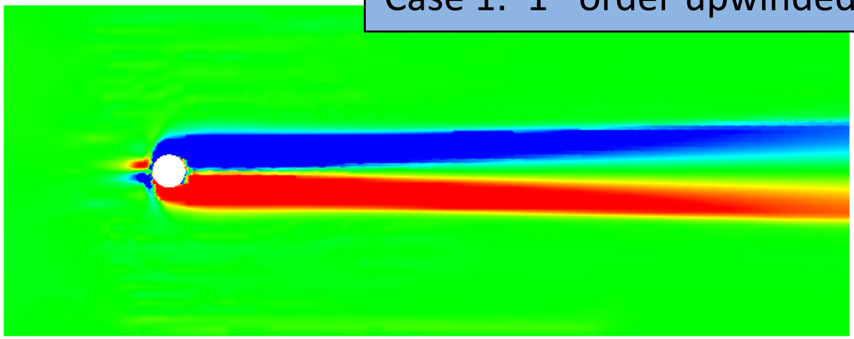


LES / 1D coupling: acoustic

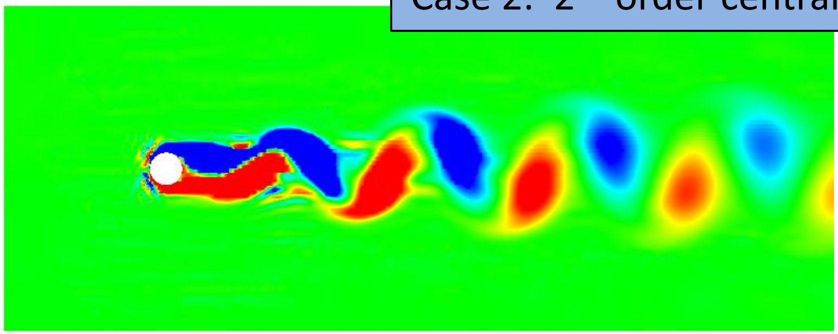
3. Modeling issues

Effect of Numerics

Case 1: 1st order upwinded



Case 2: 2nd order central



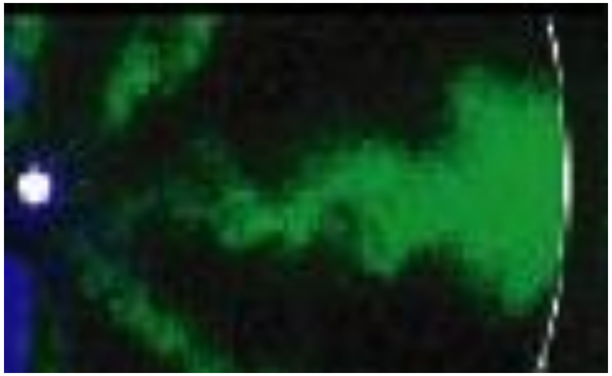
- **Case 2, which has less numerical diffusion than Case 1, results in an unsteady solution**
  - **Does not give an ensemble averaged flowfield, even when using a RANS turbulence model**
  - **A case was also run with very high resolution and 1st order upwinding, which resulted in vortex shedding, similar to Case 2**
- **The turbulence viscosity acts to destroy the smaller scales, but it also allows larger scales to exist, even if they are time-varying**

Richards et al., ASME 2014

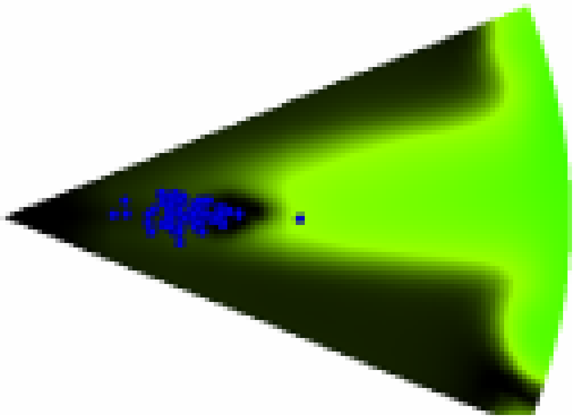
LES vs RANS

- High eddy viscosity in RANS-based models dampens non-linear velocity interactions (Rutland, IJER, 2011)
- LES models give better predictions of velocity fluctuations than RANS-based models (Liu and Haworth, Flow Turbulence Combust., 2011)
- LES predicted flow structure looks more like experimentally observed flow structure (Hu et al., SAE 2007-01-0163)

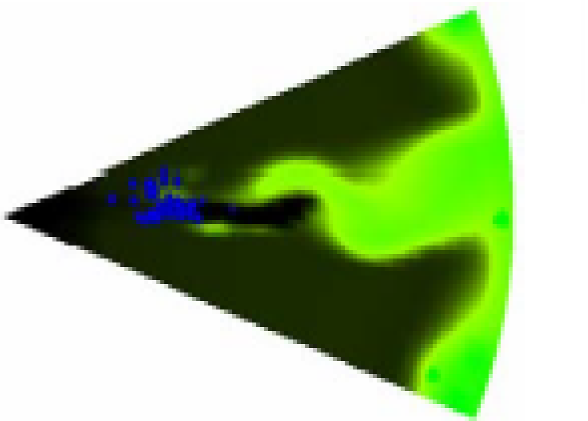
Experiments



RANS Temp.



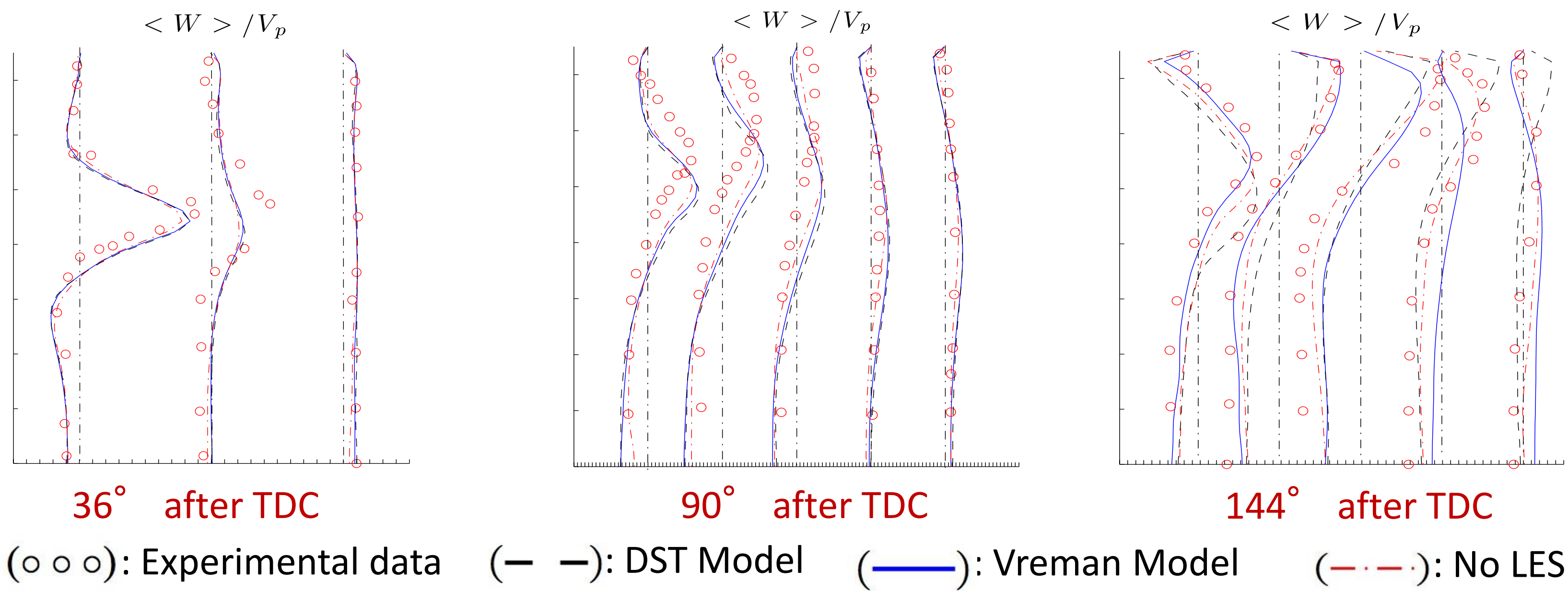
LES Temp.



Simulations of Imperial College Engine (200 RPM)

Carnegie Mellon University

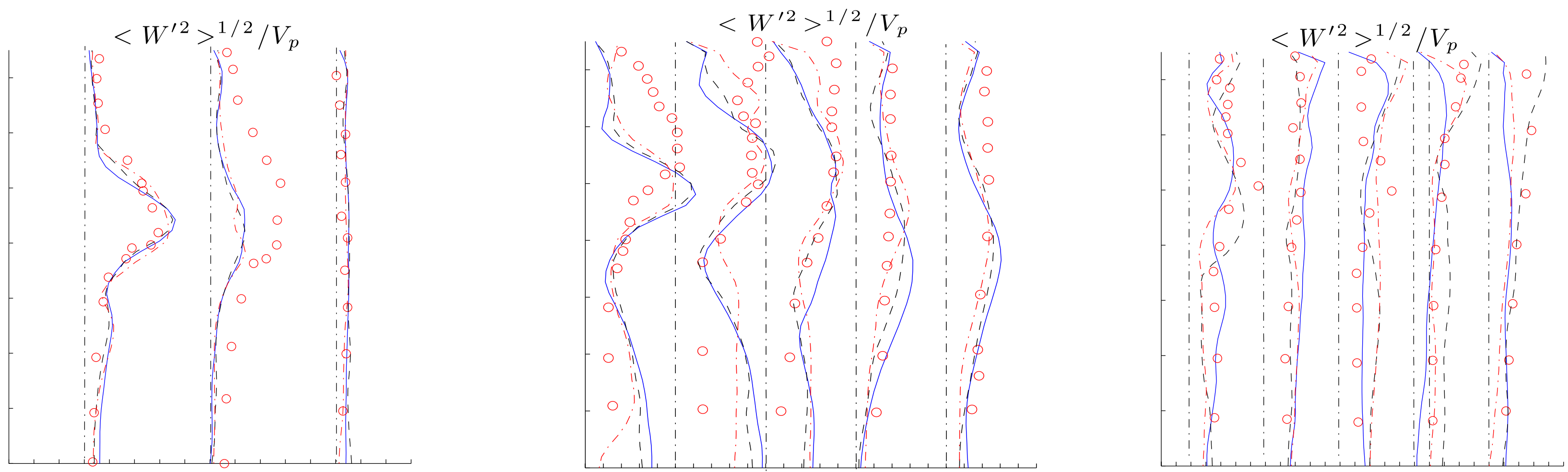
Plots of Mean Velocity



- There is hardly any different between the non-eddy viscosity DST (Pomranning &Rutland) model and eddy-viscosity Vreman model (PoF, 2004)
- Predictions are almost same when no SGS model is used indicating that SGS model does not significantly contribute to the predictions

Simulations of Imperial College Engine (200 RPM)

Plots of Root-Mean-Square (RMS) Velocity



(○ ○ ○): Experimental data (— —): DST Model (——): Vreman Model (—·—): No LES

- **LES models do not provide significantly different predictions of RMS velocity fluctuations either**

Coupling between Numerical Errors and SGS Model

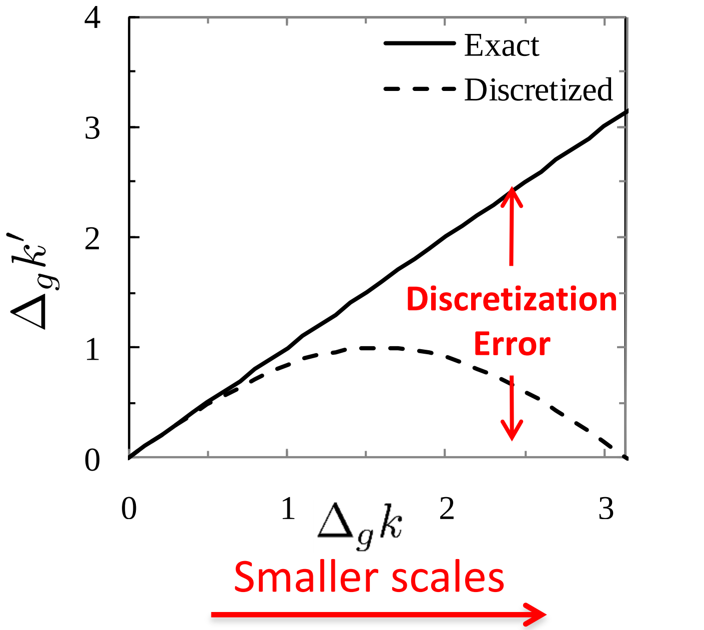
Carnegie Mellon University

Compute derivative of $f(x) = e^{\iota kx}$

Exact: $f'(x) = \iota k e^{\iota kx} = \iota k f(x)$

Numerical: $f'(x) = \iota \frac{sin(2\pi n/N)}{\Delta_g} f(x) = \iota k' f(x)$

Since $k' \neq k$, we have discretization error

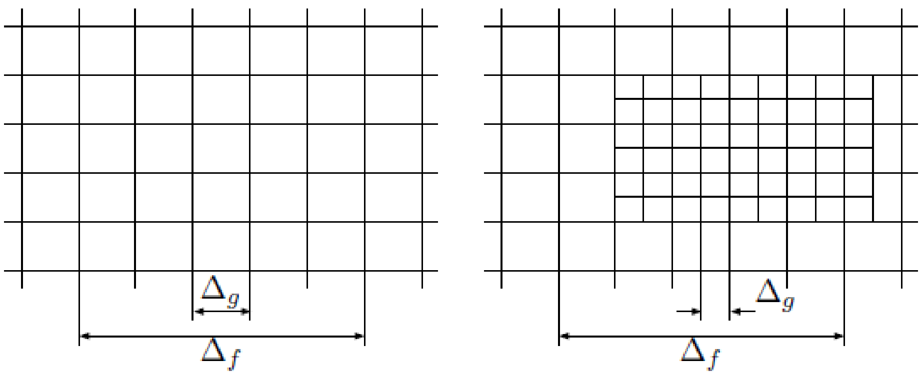


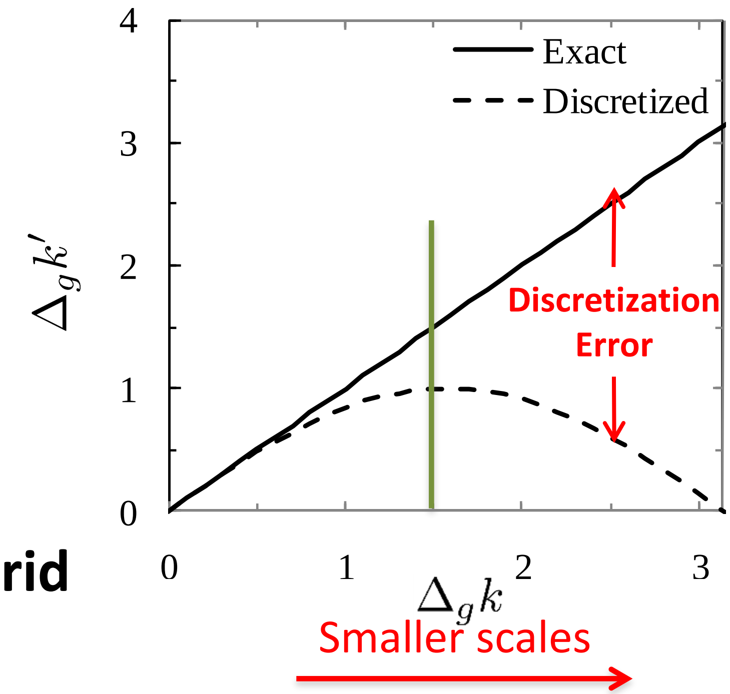
- **Coupling of ?g and k in LES**
  - **In conventional LES, grid spacing (?g) is coupled with LES resolution (k)**
  - **As ?g is reduced errors shift to higher k**
  - **Numerical errors can become larger than LES model contribution**
  - **Not easy to separate and quantify errors**
- **Choice of Numerical Scheme is Important**
  - **Most state-of-the-art codes are second-order**
  - **May not be suitable for LES if numerical dissipation is used for stabilization**

Explicit Filtering to Decouple Errors from SGS Model

Carnegie Mellon University

- **Apply an explicit filter width (Δ_f) which is larger than the grid spacing (Δ_g)**





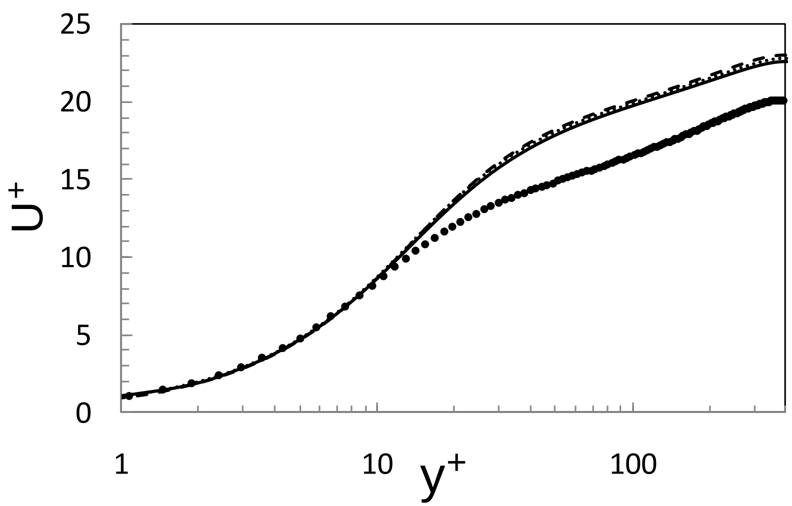
- **Discretization errors are reduced as grid is refined ($\Delta_g \to 0$) , but the effective LES resolution is kept the same (constant Δ_f)**
  - **Possible to obtain a grid-independent LES solution**
  - **Better tool for evaluating SGS models**

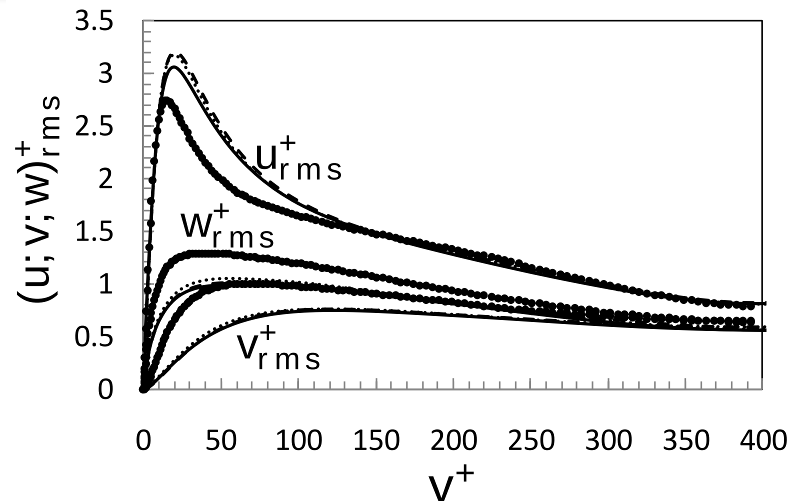
Application of Explicit Filter LES (Channel Flow)

Carnegie Mellon University

Case	N_x	N_y	N_z	Δx_e^+	Δy_e^+	Δz_e^+	FGR
Case A	64	64	64	78	0.53 - 60	39	2
Case B	96	96	96	78	0.34 - 60	39	3
Case C	128	128	128	78	0.26 - 60	39	4

—— Case A - - - Case B .... Case C symbols: DNS





- Grid-independent LES solutions are obtained for mean streamwise velocity and RMS velocity fluctuations for all filter-to-grid ratios (FGR)
- 4th-order scheme implemented on Cartesian Grid with discrete filter functions. Difficult to use in engine applications

Application of Explicit Filter LES (Channel Flow)

Carnegie Mellon University

- Differential filter of Germano, Physics of Fluids, 1986
  - Implemented in second order finite volume code (Singh and You, JCP, 2011)
  - Allows filtering on arbitrary grids
  - Filter width is controlled by change coefficient (q)

$$\bar{\phi} - \frac{\partial}{\partial x_j}\left(q \frac{\partial \bar{\phi}}{\partial x_j}\right) = \phi$$

- SGS model of Singh et al., Physics of Fluids, 2012
  - Formulated to enforce Galilean invariance for explicit-filter LES equations
  - Closure using eddy-viscosity model of Vreman, Physics of Fluids, 2004

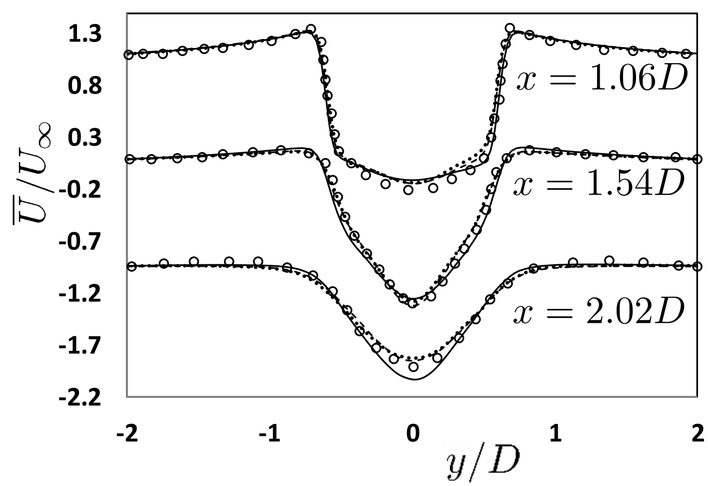
$$\tau_{ij} - \frac{1}{3}\delta_{ij}\tau_{kk} = (\bar{u}_i\bar{u}_j - \bar{\bar{u}}_i\bar{\bar{u}}_j) - 2\nu_t S_{ij}$$

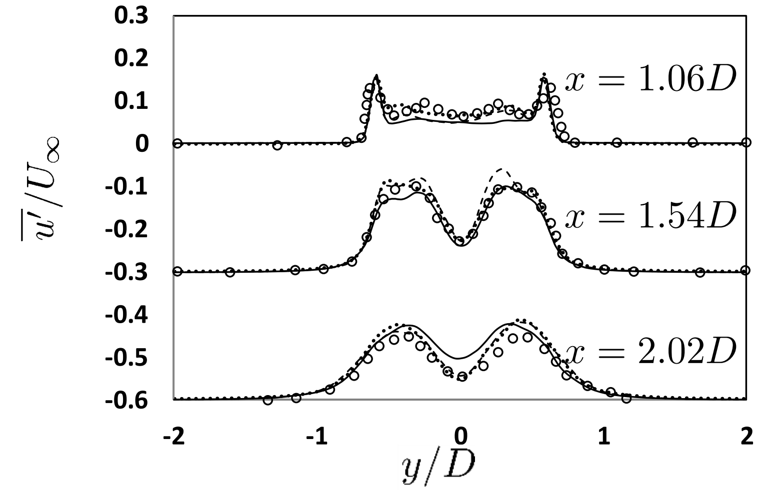
Application of Explicit Filter LES (Channel Flow)

Carnegie Mellon University

Configuration	L_x	L_y	L_z	N_I	N_J	N_{cyl}	N_K	N_{total}	FGR	q
A	-20D to 20D	-25D to 25D	πD	570	136	136	54	4 186 080	1.50	0.30
B	-20D to 20D	-25D to 25D	πD	674	160	160	64	6 901 760	1.75	0.40
C	-20D to 20D	-25D to 25D	πD	754	180	180	72	9 771 840	2.00	0.47

—— Case A - - - Case B .... Case C symbols: DNS





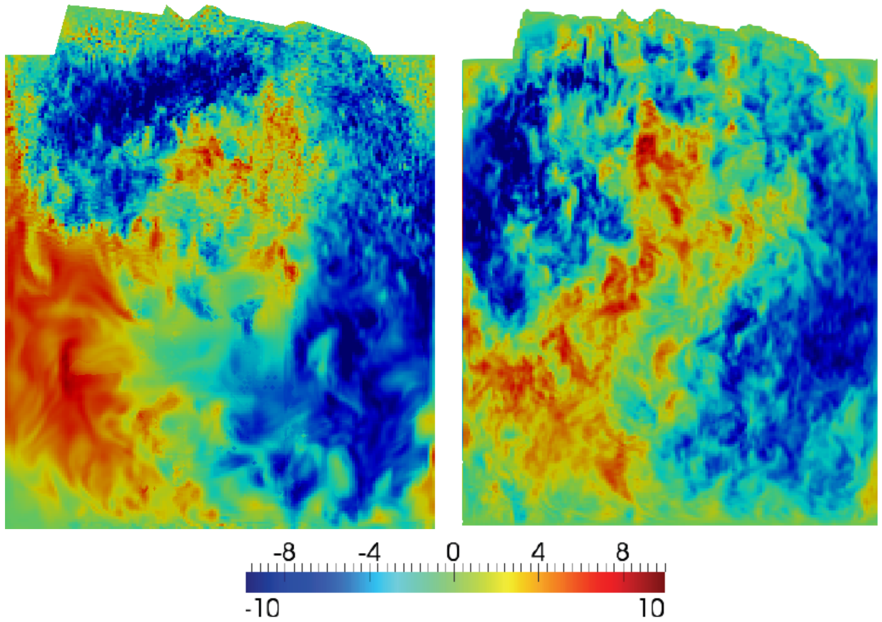
- **Streamwise mean velocity (left) and velocity fluctuations (right) are nearly grid independent for the two finer grids**
- **Differential filter can be applied in ICE simulations**

Duisburg Groups[1]

UNIVERSITÄT DUISBURG ESSEN

Two modelling approaches (A. Kempf):

OpenFOAM PsiPhi

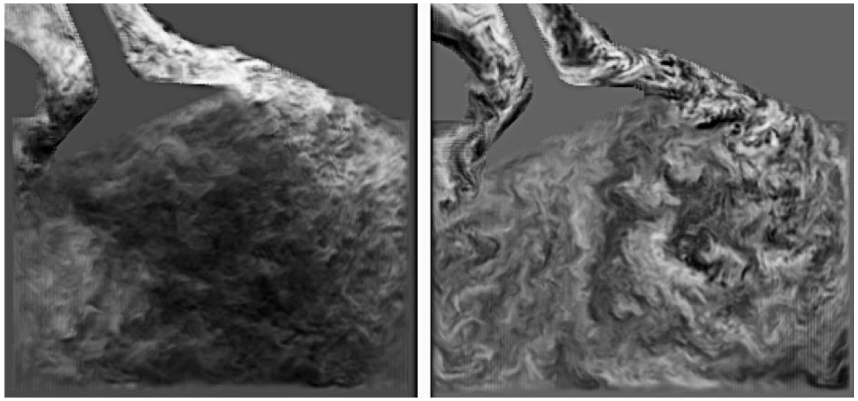


- **Higher resolution for OpenFOAM**
- **PsiPhi less diffusive**
  - **Higher temporal accuracy and a mapping free strategy**
  - **Immersed boundaries for moving objects**
- **Simulation-to-simulation comparison and validation against Duisburg optical engine[2]:**
  - **PsiPhi: density-based, explicit, structured grids with immersed boundaries**
  - **OpenFOAM: pressure-based, implicit, unstructured moving grids**

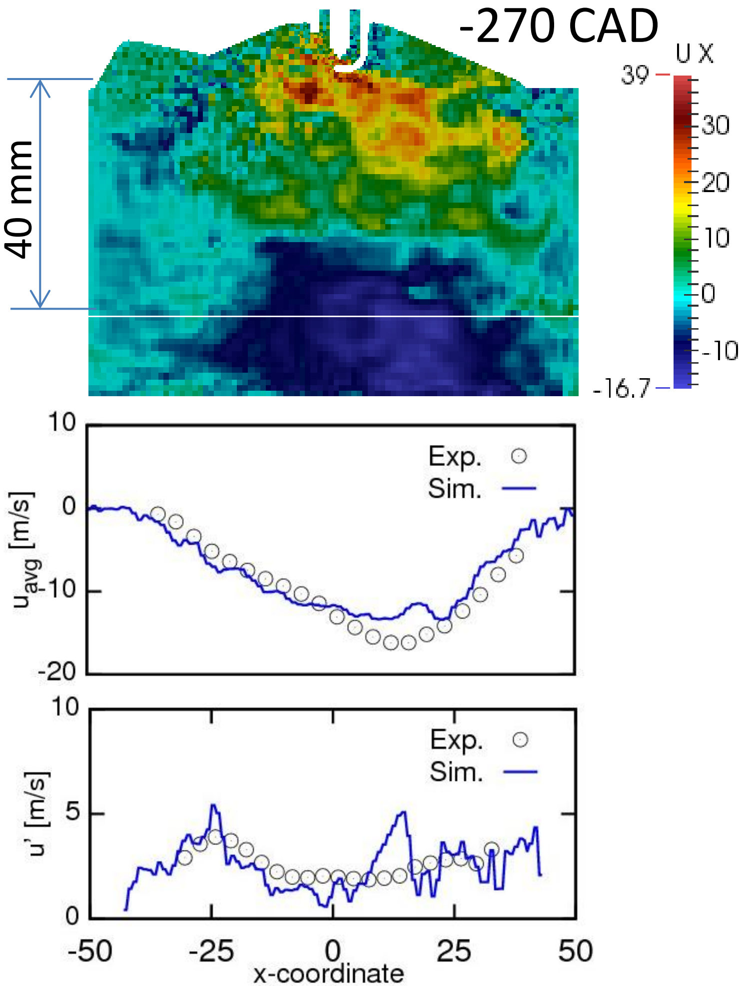
[1]Chair of Fluid Dynamics and chair of Reactive Flows, University of Duisburg-Essen, Germany
[2]Nguyen, Janas, Lucchini., D'Errico, Kaiser, Kempf, LES of Flow Processes in an SI Engine using Two Approaches: OpenFoam and PsiPhi (SAE Paper 2014-01-1121).

Spatial resolution

- **Grid sensitivity studies with PsiPhi on a 0.3mm, 0.5mm, 1 mm grid**
  - **Good agreement among the simulations and experiment**
- **Multi-cycle simulations with OpenFOAM, (0.125 mm in the valve gap, 1 mm inside the cylinder, 2 mm inside the manifolds)**

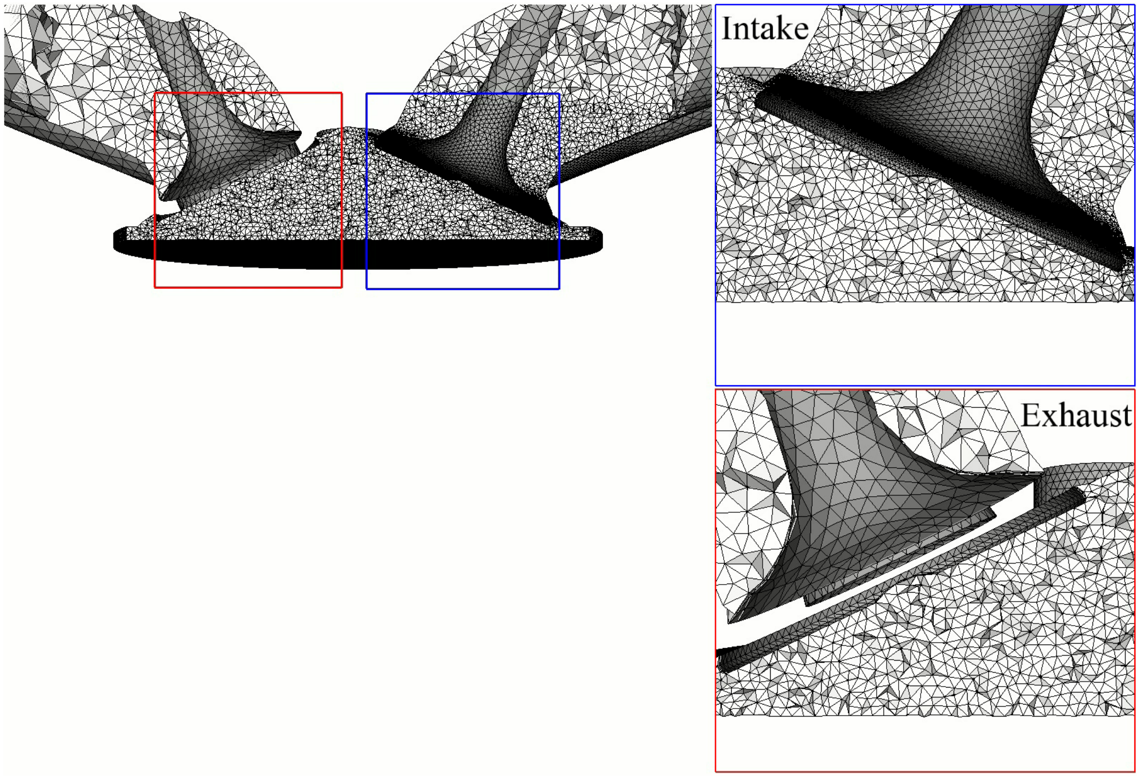


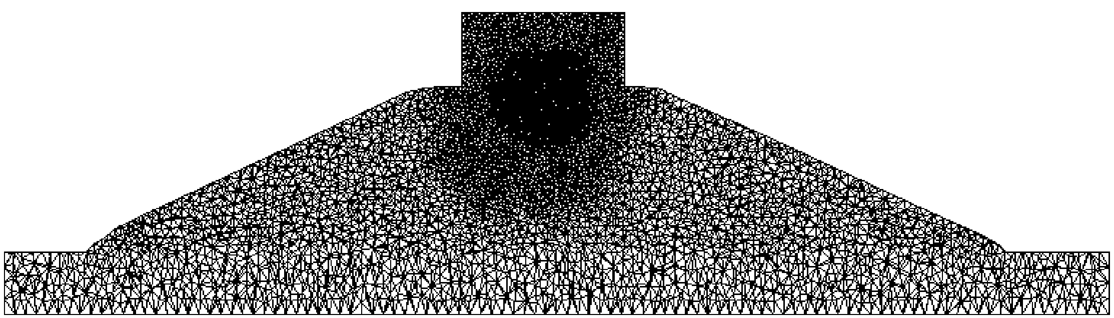
High-Resolution LES of the Darmstadt engine (0.3 mm) with PsiPhi , (T. Nguyen)



Multi-cycle simulation (5 Cycles), OpenFOAM, mean velocity and fluctuations compared to PIV, Darmstadt engine,(P. Janas)

Spatial resolution





IFP Energies nouvelles:

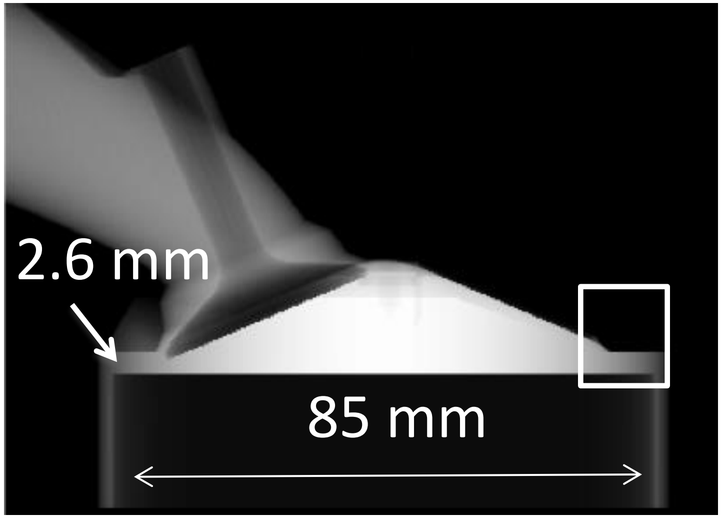
- From 1 to 10 M cells
- 0.1 mm: typical size around the valve seat
- 0.5 mm: typical mesh size in the cylinder
  - 0.2 mm: around the spark plug

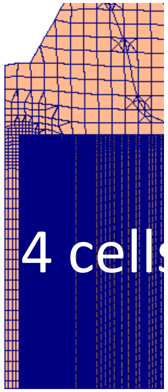
Politecnico di Milano:

- Mesh size: 0.125 mm (valve region) to 2 mm (intake and exhaust ports).

Modeling of the crevice volume

- Large crevice volume between the piston-skirt and cylinder liner
  - Up to 15% of the top dead center volume (excluding piston expansion)
- Fresh air/fuel mixture trapped in crevice volume
  - 50% trapped at TDC is possible
  - Not available for combustion
- Crevice volume in simulation
  - Reduces the peak pressure by 10 bar



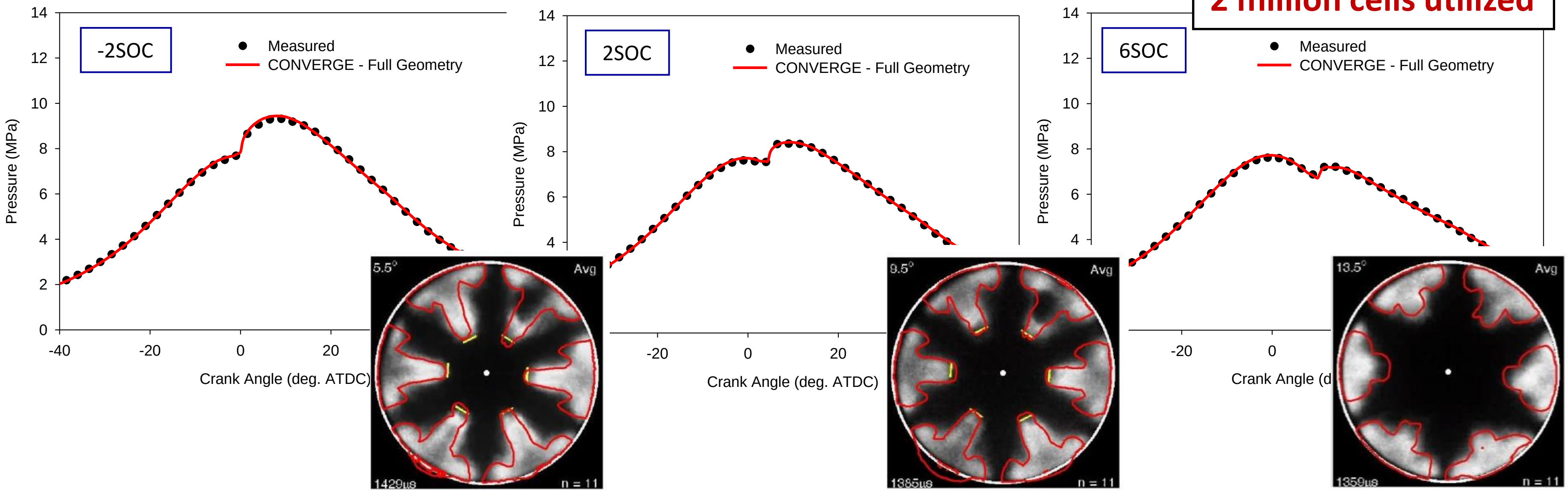


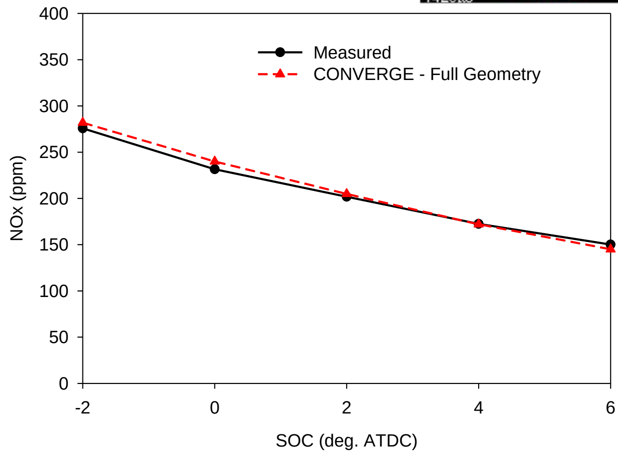
Engine grid with crevice volume, Darmstadt engine, (Janas/Nguyen)

Combustion: grid-convergent methodology

CONVERGENT SCIENCE

2 million cells utilized





OH* Chemiluminescence vs. OH iso-surface, 7.5 degrees after SOC

The grid-convergent methodology results in excellent agreement with global combustion behavior as well as flame lift-off length and flame location

*Senecal et al. ASME 2013

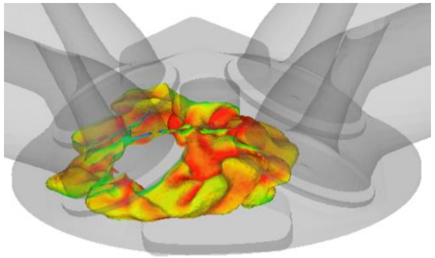
Collaboration with Caterpillar Inc. and Sandia National Laboratories

Turbulent Combustion Model: CFM-LES

- **FSD (Flame Surface Density) transport equation**
  - **Adaptation from RANS to LES [Richard et al., Proc. Combust. Inst. 2007]**

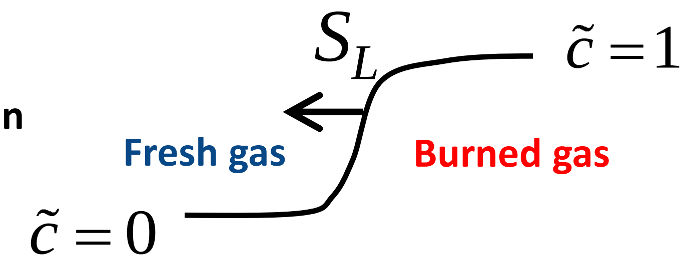
$$
\left.
\begin{aligned}
\frac{\partial \overline{\Sigma}_{\tilde{c}}}{\partial t} + \nabla \cdot \left(\tilde{\mathbf{u}} \overline{\Sigma}_{\tilde{c}}\right) = {} & \left(\nabla \cdot \tilde{\mathbf{u}} - \bar{\mathbf{n}}\bar{\mathbf{n}} : \nabla \tilde{\mathbf{u}}\right) \overline{\Sigma}_{\tilde{c}} \\
& - \nabla \cdot \left(S_d \bar{\mathbf{n}} \overline{\Sigma}_{\tilde{c}}\right) + S_d \nabla \cdot \bar{\mathbf{n}} \overline{\Sigma}_{\tilde{c}}
\end{aligned}
\right\} \text{Resolved contributions}
$$

$$
\left.
\begin{aligned}
& + \nabla \cdot \left(\frac{\nu_t}{Sc_t} \nabla \overline{\Sigma}_{\tilde{c}} \right) \\
& + \Gamma \left(\frac{u'}{S_l}, \frac{\Delta}{\delta_l} \right) \frac{u'}{\Delta} \overline{\Sigma}_{\tilde{c}} \\
& + \beta_c S_l \frac{c^* - \bar{c}}{\bar{c}(1 - \bar{c})} \left(\overline{\Sigma}_{\tilde{c}} - \overline{\Sigma}_{\tilde{c}}^{lam} \right) \overline{\Sigma}_{\tilde{c}}
\end{aligned}
\right\} \text{SGS contributions (need modelling)}
$$



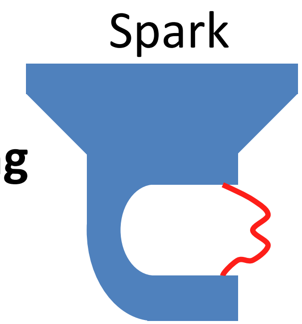
where $\tilde{c} = 1 - \widetilde{Y}_{fuel} / \widetilde{Y}_{T\,fuel}$ → **fuel mass fraction in the fresh gases**

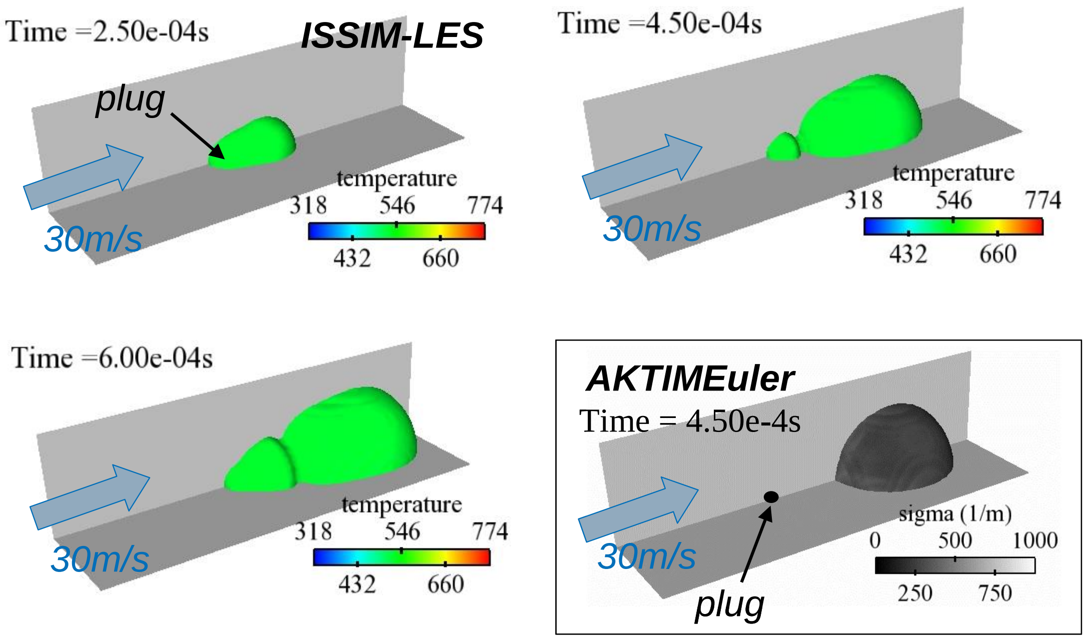
$\widetilde{Y}_{fuel}$ ↓ **fuel mass fraction**



Turbulent Combustion Model: ignition with ISSIM-LES[1]

- ISSIM (Imposed Stretch Spark Ignition Model)
- Description of the electrical circuit
- Use of the FSD transport equation from spark timing to quenching
  - Account for local convection and wrinkling
  - Simulate multiple-ignitions

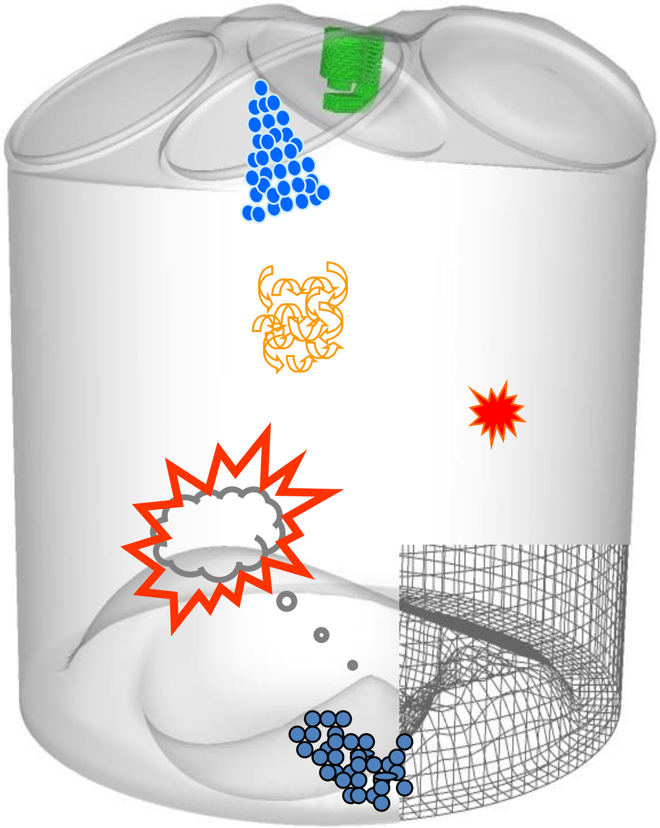




[3] O. Colin and K. Truffin. Proc. Combust. Inst. 33(2) (2011)

Conclusion

- Today, many engine codes exist with different approaches
- Work and development within ECN?
  - Comparison and Validation: Topic 4.3
  - Comparison between codes on reference ECN database?



Contributions

1. Meshing strategy

Cecile Pera, cecile.pera@ifpen.fr AVBP, IFP-C3D
Tommaso Lucchini, tommaso.lucchini@polimi.it Lib-ICE
Andreas Kempf, andreas.kempf@uni-due.de OpenFOAM, PsiPhi
Kelly Senecal, senecal@convergecfd.com Converge

2. Boundary conditions and methodology

Cecile Pera, cecile.pera@ifpen.fr AVBP, IFP-C3D
Tommaso Lucchini, tommaso.lucchini@polimi.it Lib-ICE
Andreas Kempf, andreas.kempf@uni-due.de OpenFOAM, PsiPhi
Stefano Fontanesi, stefano.fontanesi@unimore.it star-cd

3. Modeling issues

Cecile Pera, cecile.pera@ifpen.fr AVBP, IFP-C3D
Andreas Kempf, andreas.kempf@uni-due.de OpenFOAM, PsiPhi
Kelly Senecal, senecal@convergecfd.com Converge
Satbir Singh, satbirs@andrew.cmu.edu CMU

Please, come to the LES4ICE meeting

LES for Internal Combustion Engine Flows

Where: Rueil-Malmaison, France
When: 4-5 December 2014

The AVBP code

- Compressible Navier-Stokes equations
- Finite volume, cell-vertex
- Explicit
- Unstructured meshes

- Lax-Wendroff (centered, 2nd order)
- Smagorinsky turbulent SGS model
- NSCBC Boundary condtions
  - Linear Relaxation Model
- Moving grid management