CFD combustion simulations of a two-valve Diesel engine

Luca Vargiu
FPT Industrial
Basic Technologies – Thermo & Fluid Dynamics

Milan

22 February 2018

Outline

- Objective
- Introduction
- Meshing
- Simulation setup and combustion model
- Results
- Conclusions

Objective

- Run combustion simulations of a two valve Diesel engine
- Develop a methodology for the meshing of a full 360° cylinder geometry in OpenFoam
- Validate the CFD model over experiments with focus on emissions

Introduction

- CFD combustion simulations of diesel engines are usually carried out on a cylinder sector to:
  - Suite the symmetry of the combustion chamber
  - Save time and computational resources

- The combustion chamber in a two-valve engine is not symmetric. A proper mesh is then required to:
  - Simulate the full 360° cylinder
  - Correctly represent the eccentricity of the combustion chamber with respect to the piston
  - Include geometric features of the cylinder head, as the injector nozzle and valve seats

- The so-defined mesh and the entire CFD model must be validated vs. experimental data

Meshing

Mesh generation workflow

The building of a fully 360° mesh may be achieved with the following steps:

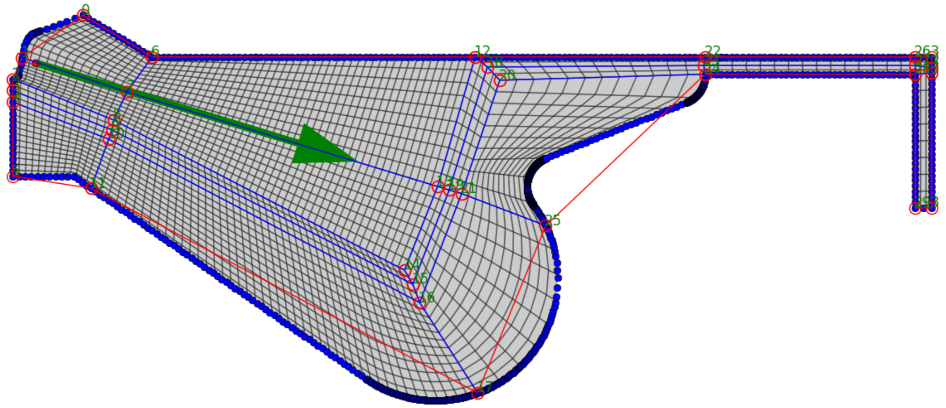
1. A symmetric mesh of the combustion chamber is created
2. A correction is applied to ensure mesh usability
3. A rigid translation is applied to the bowl to meet the user-defined eccentricity value
4. Valve seats and injector nozzle are included in the full 360° mesh

Meshing

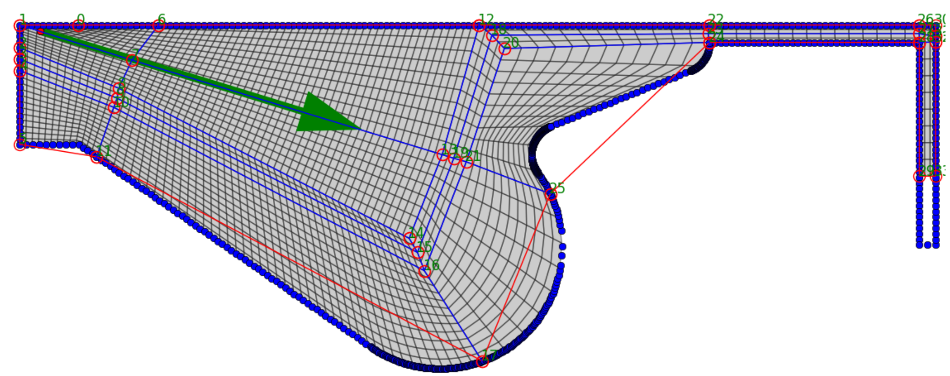
Step1: creation of a symmetric mesh with the DCC Mesh Generator tool

Two approaches are available:

1. Injector and valve seats are included in the 2D profile → the final 3D mesh does not represent the actual asymmetric head geometry



2. Flat head profile→ allows the modelling of the actual head geometry



Since the 3D mesh is built by revolution, a sector of amplitude 360° is used

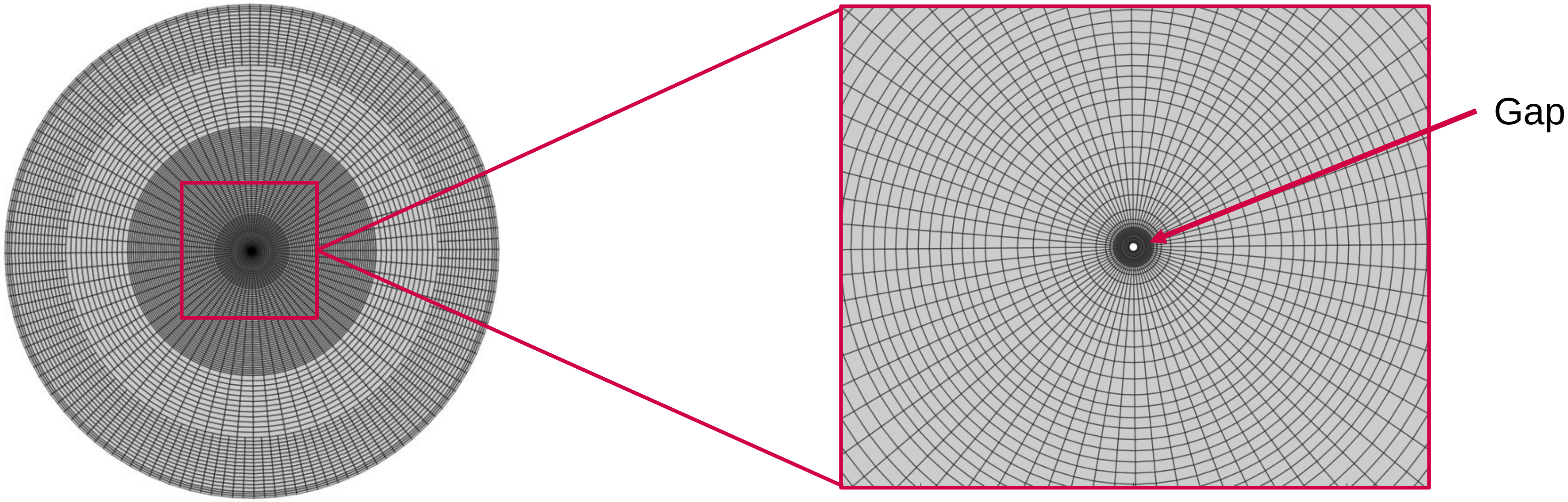
Meshing

Step 2: mesh correction

A full 360° symmetric mesh is created:

- A hole is present in the centre of the bowl, where the symmetry condition is usually applied
- Bowl eccentricity is not obtained yet

The *closeAxialHole* utility must be used to fill the gap in the centre of the bowl

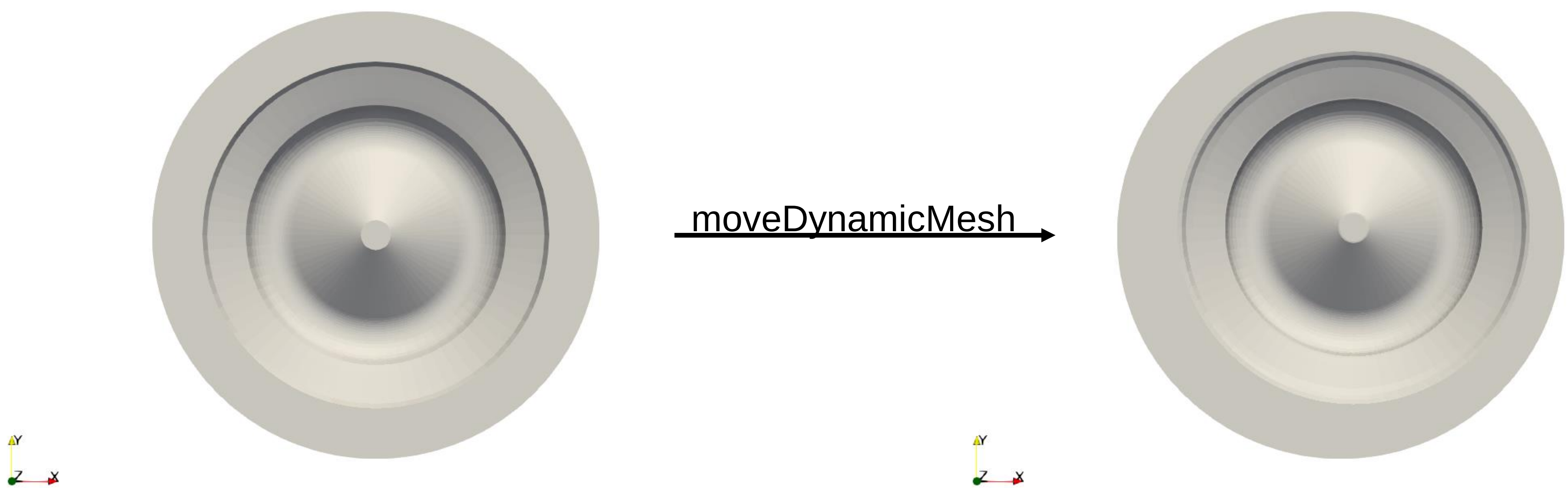


Meshing

Step 3: bowl eccentricity

The eccentricity is applied to the bowl:

- Geometric input values are read from the *meshParameters* file
- The *moveDynamicMesh* utility is executed to move the bowl

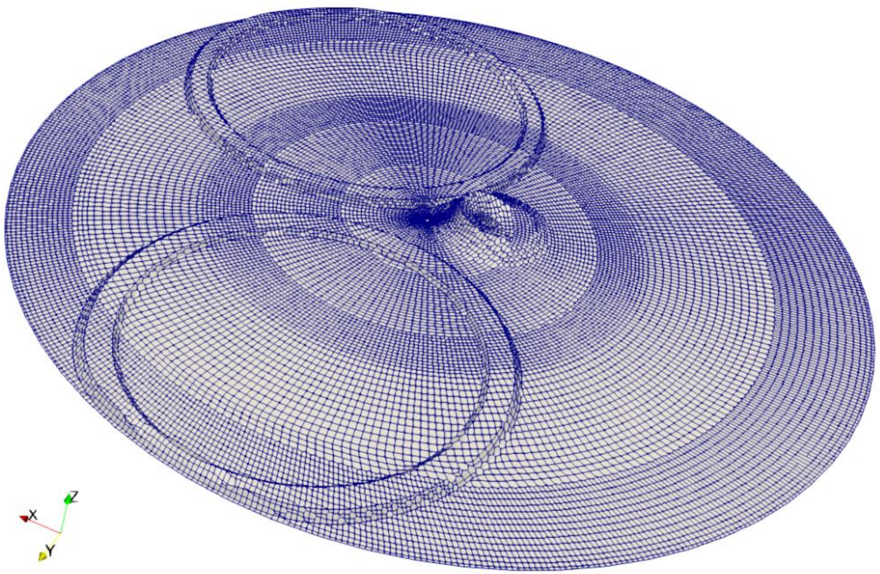


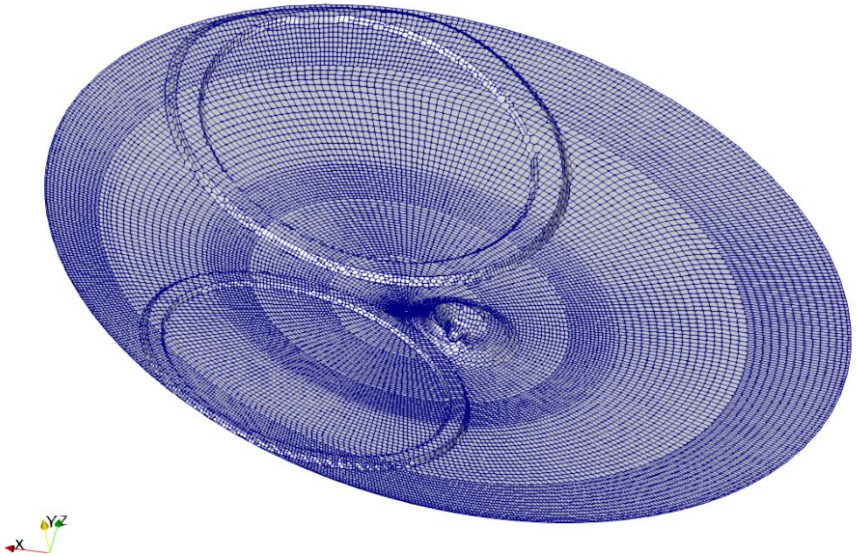
Meshing

Step 4: valve seats and injector modelling

The geometrical features of the head can be added to the already-available 360° mesh:

- An .stl file of the head geometry is used
- The provided surface is then snapped to the existing flat head





Simulation setup

Combustion model: Representative Interactive Flamelets

Combustion is simulated with the single Representative Interactive Flamelets model (RIF)

- Turbulence time scale are much larger than chemical ones
- Chemical reactions occur within an undisturbed sheet, modelled with diffusion flames (flamelets)
- All reacting scalars and their temporal evolution only relate to the mixture fraction Z: transport equations are written for Z and its variance (including spray evaporation terms)
- Flamelet equations for species mass fraction and sensible enthalpy are solved in the 1D mixture fraction space
- Coupling between turbulence and chemistry is considered thanks to a dissipation rate term, function of turbulent kinetic energy, dissipation rate and Z variance

Simulation setup

Combustion model: Representative Interactive Flamelets

Summarizing:

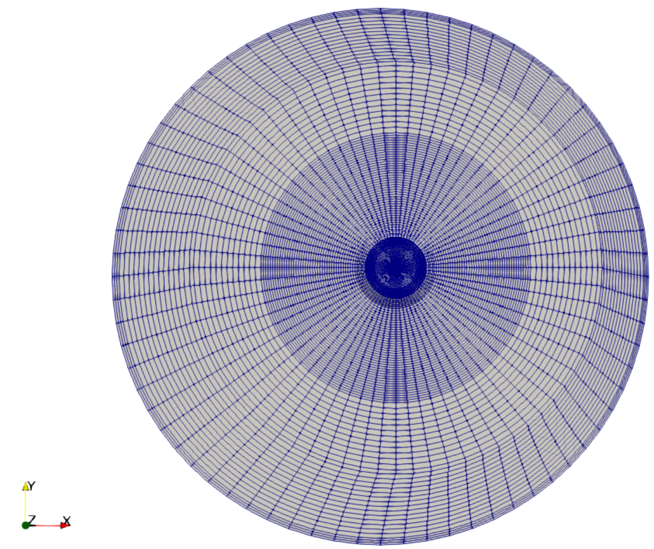
1. Scalar dissipation rate values are passed from the CFD domain to the flamelet equations

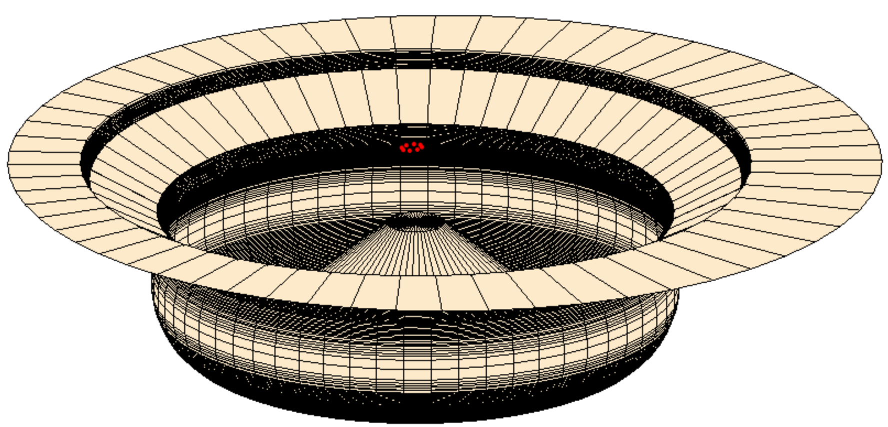
2. Flamelet equations are solved → chemical composition in the Z space

3. Chemical composition in the CFD domain is obtained, given:
   - Species mass fractions in the Z space
   - Z and its variance

Simulation setup

- Simulation were carried out between Intake Valve Closure and Exhaust Valve Opening
- A flat head with injector nozzle geometry was used to reduce the simulation setup time
- Spray was modelled with the Kelvin-Helmotz Rayleigh-Taylor approach (blob injector)
- Injector nozzle holes and spray directions were accurately defined in the 3D space
- The single RIF model was used: one flamelet per injection





Results

CFD model validation vs. experimental data

The following conditions were analysed:

- Engine speed variations at fixed load and hardware
- Hardware variations at fixed load and engine speed:
  - Fixed injector and spray angle
  - Nozzle Tip Protrusion (NTP) variations
  - Swirl level: low vs. high swirl

In the following, a comparison between CFD and experiments is shown with focus on:

  - Specific soot emissions
  - NOx emissions

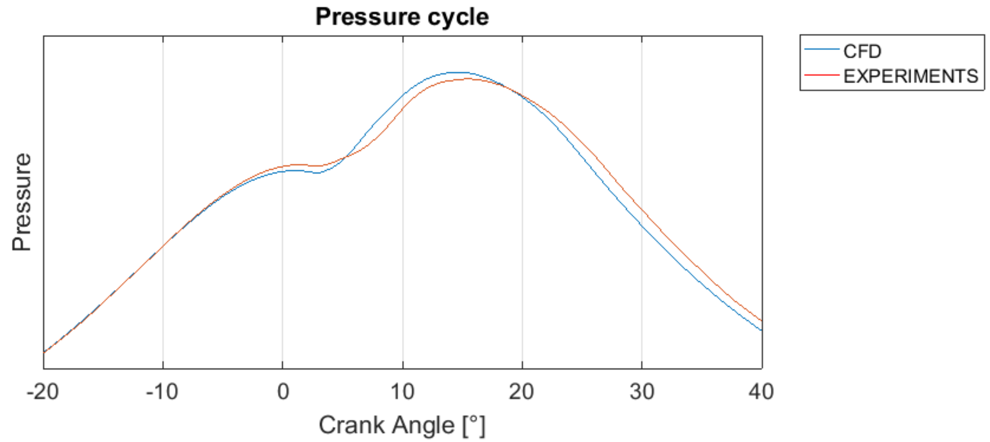
- CFD specific quantities are ‘Indicated’, while experimental ones are ‘Brake Specific quantities’

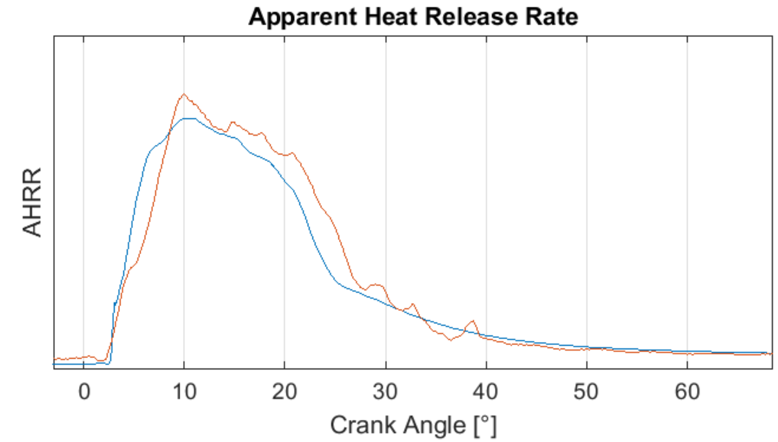
<u>Note: all the results are normalized over a reference condition</u>

Results

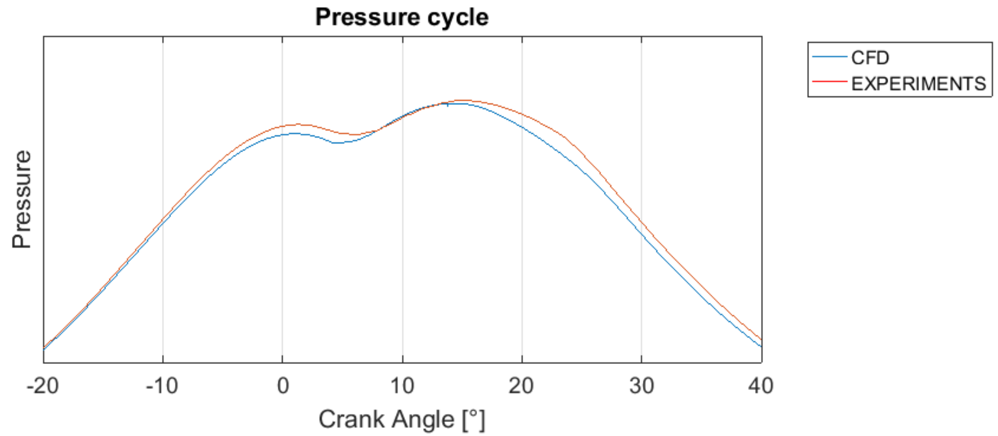
CFD model validation: fixed load and hardware configuration

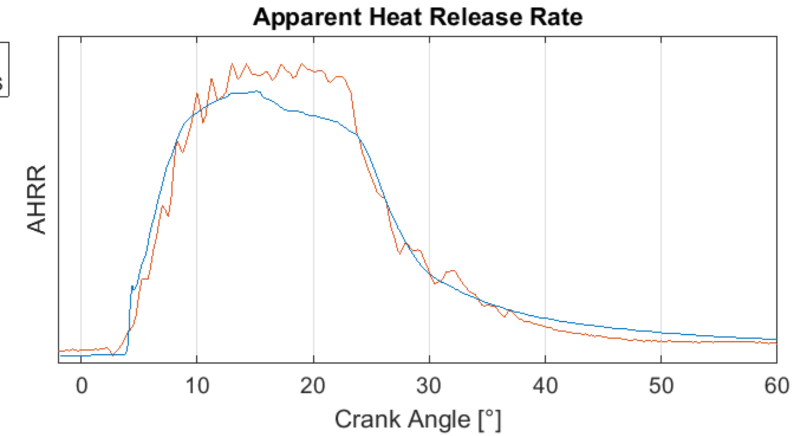
- Point @ 1400RPM:





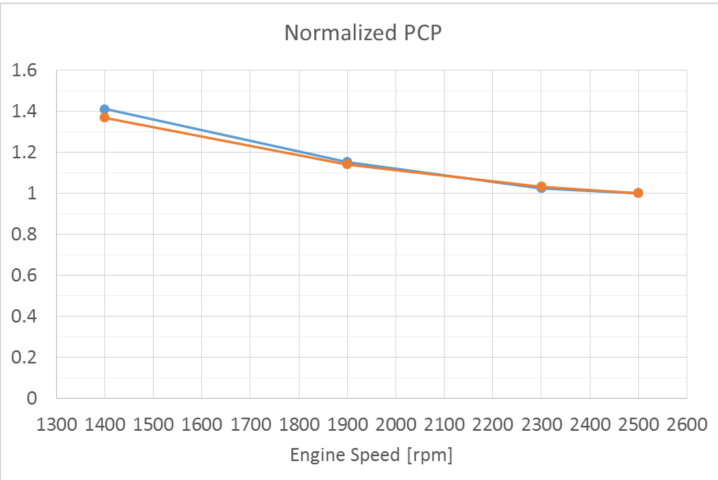
- Point @ 1900RPM:

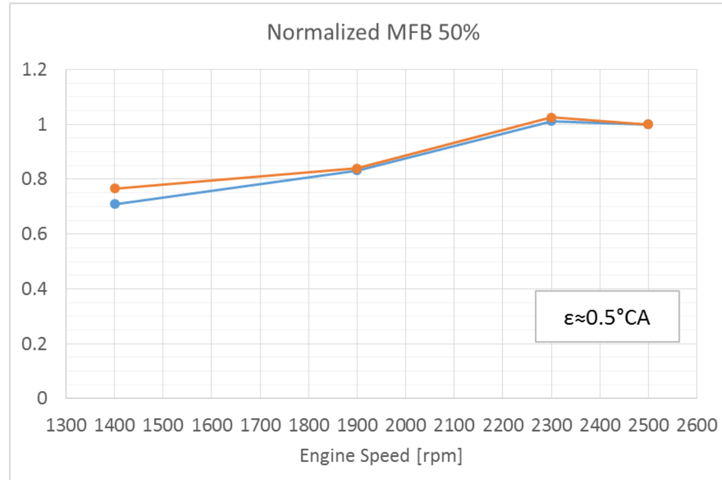


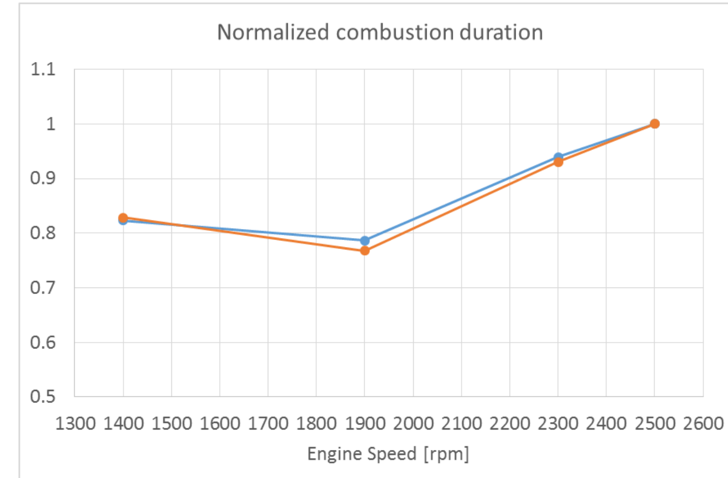


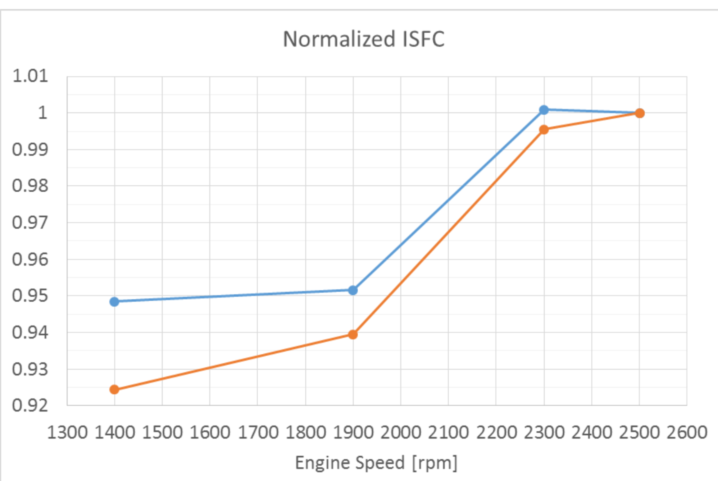
Results

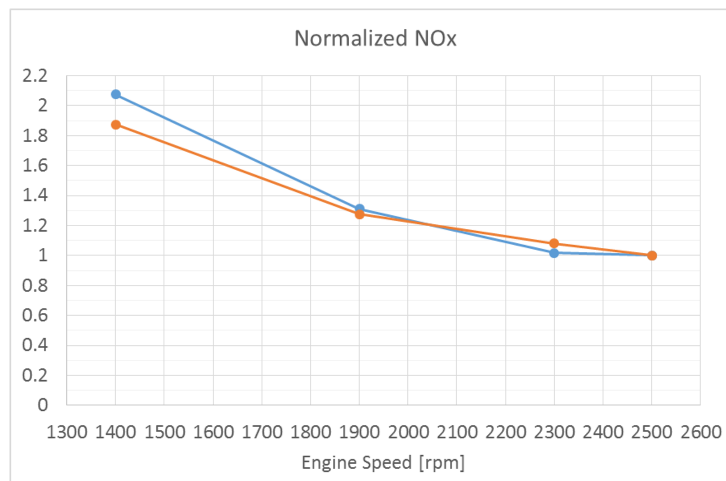
CFD model validation: fixed load and hardware configuration

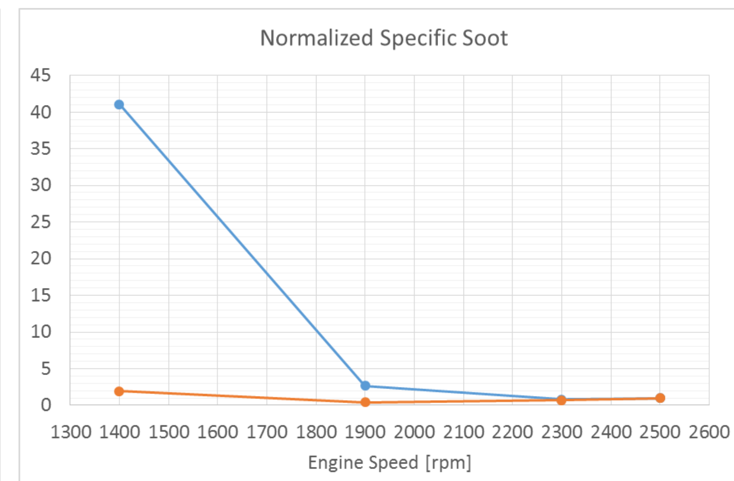












CFD EXP

Results

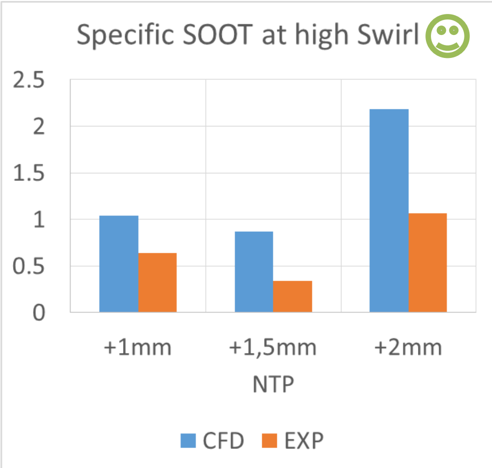
CFD model validation: fixed load and hardware configuration

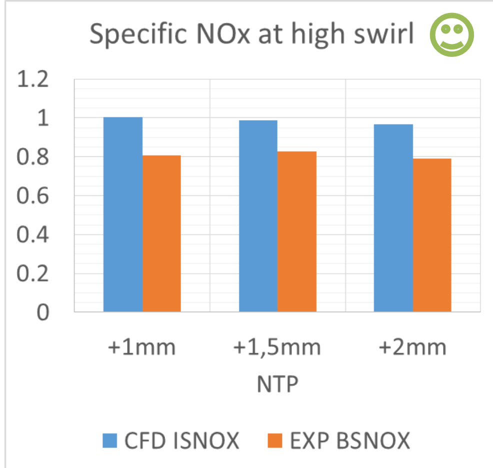
- Peak Cylinder Pressure is evaluated correctly
- The centre of combustion is accurately predicted by the CFD model: for all points, it differs less than 1°CA from the experimental value (average error is 0.5°CA)
- Specific Fuel Consumption variation are predicted with accuracy 1-2%
- NOx emissions trend is captured
- Soot at 1400RPM is overestimated and point at 1900RPM is out of trend. However, the information that a low- and an high-soot zone are present is correct

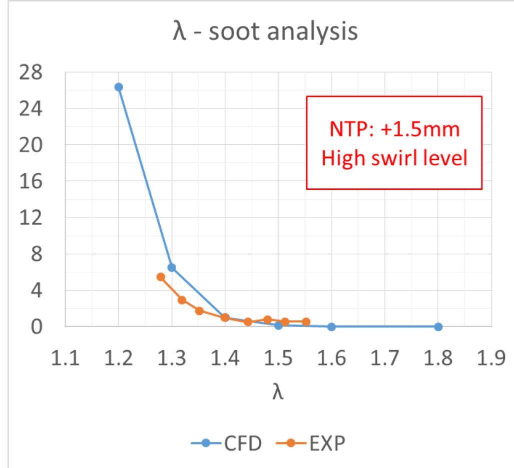
Results

CFD model validation: injector protrusion variations at fixed swirl conditions and engine speed

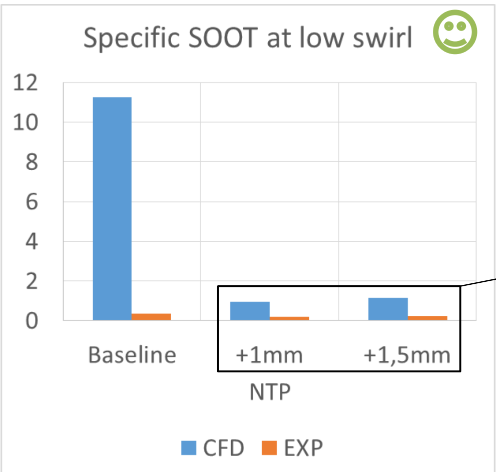
- High swirl case:

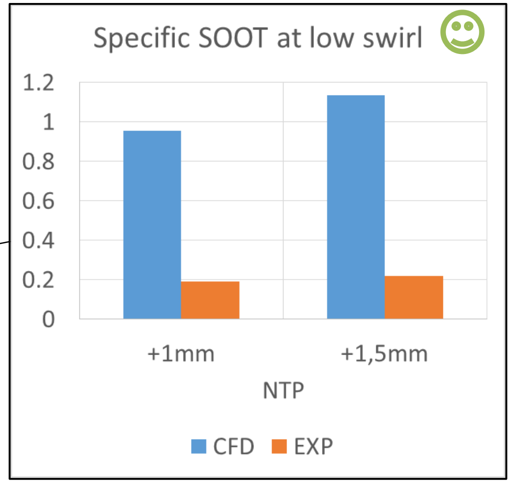


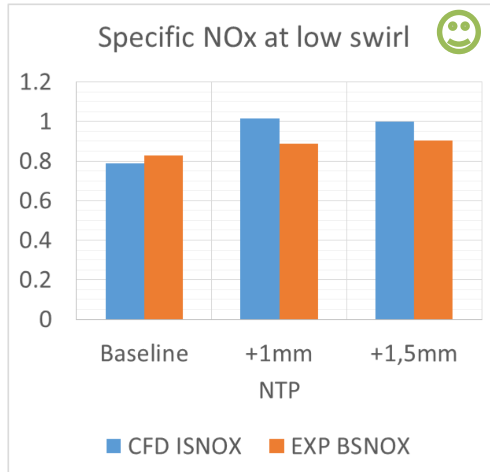




- Low swirl case:







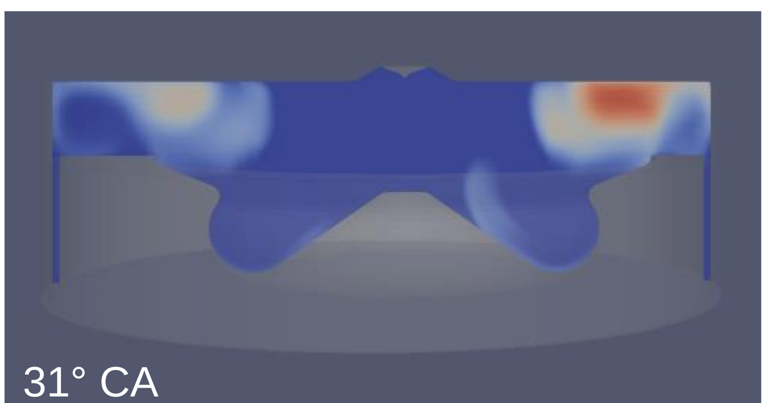
Results

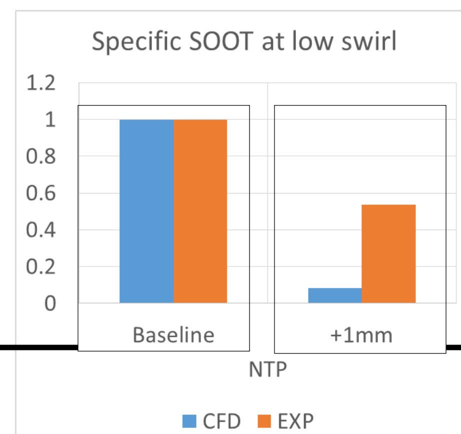
CFD model validation: injector protrusion variations at fixed swirl conditions and engine speed

Looking at the soot visualizations in the low swirl conditions, the simulation provides the correct information:

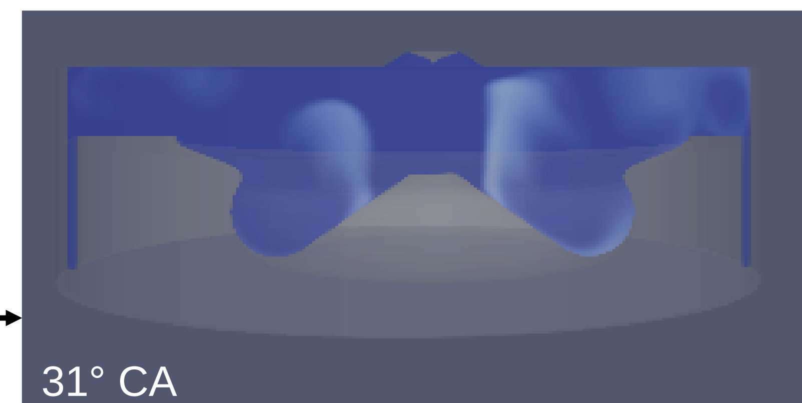
- High soot concentrations in the upper part of the combustion chamber due to a small injector protrusion (baseline case)
- Soot reduction with an increased +1mm protrusion

Soot mass fraction: baseline protrusion





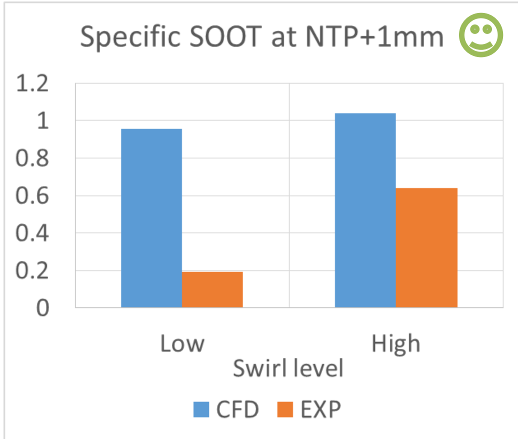
Soot mass fraction: +1mm protrusion

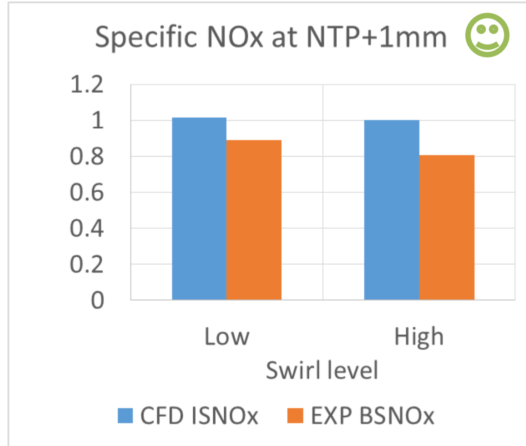


Results

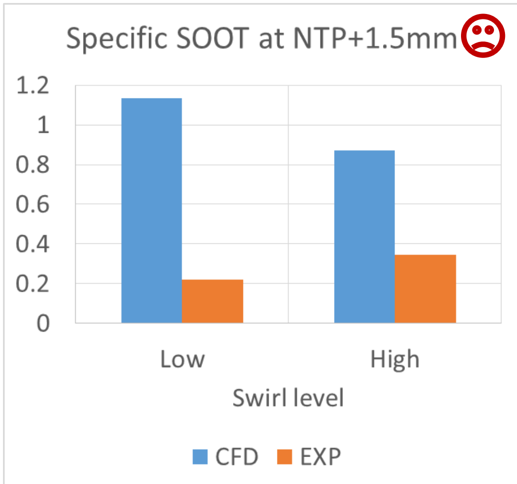
CFD model validation: swirl level variation at fixed injector protrusion and engine speed

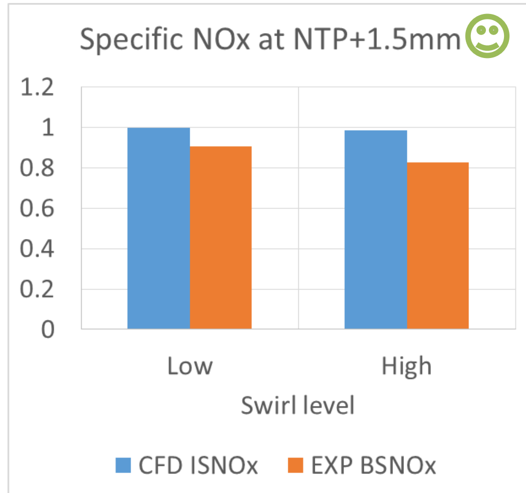
- Nozzle Tip Protrusion: Base +1mm:





- Nozzle Tip Protrusion: Base +1.5mm:





Results

CFD model validation at fixed engine speed

The previous results show that:

- Specific NOx emissions are in good agreement with measurements: variations are small and, if not, the model captures the trend
- Soot variations are evaluated correctly (qualitatively):
  - The CFD overestimates the soot variations
  - A huge overshoot is visible for the baseline protrusion – low swirl case
  - Trends are captured well in 3 out of 4 cases
- The λ-soot analysis suggests that the CFD model is able to detect the critical λ value related to soot increasing
- Overall, the code is reliable in predicting pollutants variations when hardware configurations are introduced

Conclusions

- A methodology for simulating combustion in a two-valve diesel engine was investigated
- OpenFoam, along with LibICE, offer a tool that enables:
  - The meshing of a 360° cylinder sector with an eccentric bowl
  - The accurate representation of the cylinder head with valve seats and injector
- The validation of the CFD model vs. experiments was shown at:
  - Fixed hardware, load and varying engine speed
  - Fixed engine speed, load and varying swirl levels/injector protrusion
- Performance and combustion phasing is predicted correctly

Conclusions

Focusing on the emissions results:

- The predicted NOx trends are in good agreement with the experimental data
- Soot variations from case to case are overestimated, however the information is overall qualitatively consistent with experiments
- Hypothetically, the same conclusion may be reached in the choice of the best hardware configuration when either looking at numerical or experimental data
- Not modelling the injector nozzle and valve seats (instead using a flat head with simplified injector nozzle profile) do not seem to undermine the accuracy of the results

<u>Overall, the CFD model provides a reliable prediction of pollutants when investigating different hardware configurations</u>

Thanks for your attention