CFD SIMULATIONS FOR DEVELOPMENT OF HIGH PERFORMANCE MARINE DIESEL

Author: Eng. M. Magni
Seatek Tech. Dept.

11 July 2011

About Seatek S.p.a.

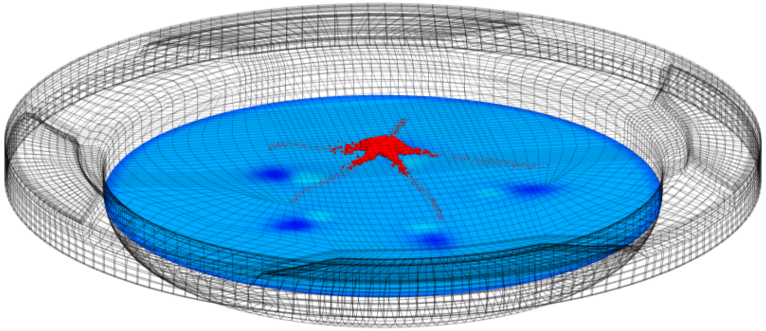
- Founded in 1986
- Specialized in marine engines
- 2 type of engine produced: L6 and V12

Introduction

Type of application:

- Fast interceptor craft
- Fast patrol boat
- High performance boat (pleasure)
- Fast boat (motorsport)

1. Analysis of combustion process



1. Combustion process

➢ Main TARGETS of CR application on Seatek engine

- Obtain the ***same*** performances of the previus mechanically controlled version
- Meet the new marine emission regulation with ***no*** EGR and ***no*** DPF and SCR systems:

IMO; EU; EPA (Blue sky)

ISO 8178 cycle E3 and E5

NOx + HC	→	5,8 gr/kWh	(EU)
PM	→	0,12 gr/kWh	(EU)

1. Combustion process

WITH OpenFOAM WE…..

- …would be able to simulate the whole engine cycle, and it's been validated by experimental data.

Target of simulation

- Investigation on Combustion process:
  - Injector geometries
  - Injection pressure
  - Swirl ratio
  - Bowl geometry

1. Combustion process

- Standard characteristics of Seatek 880CR

	Value
Holes number	5
Swirl Number	2
Hole diameter	0.27 mm
Holes angle	145°
Rail pressure	150 MPa

1. Combustion process

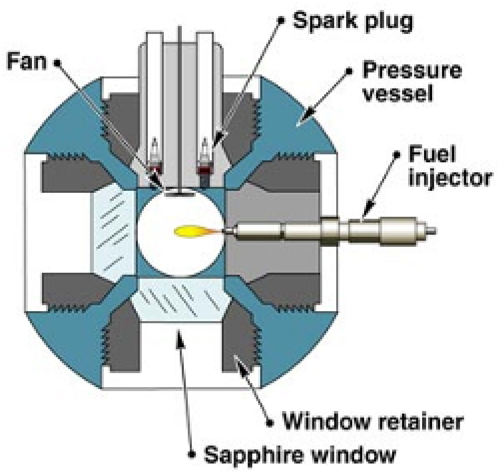
1. Model validation
   - Spray validation
   - Engine data validation
2. Optimization of combustion process
   - Injection nozzle geometry
   - Effect of Swirl coefficient
   - Injection pressure
   - Piston bowl geometry

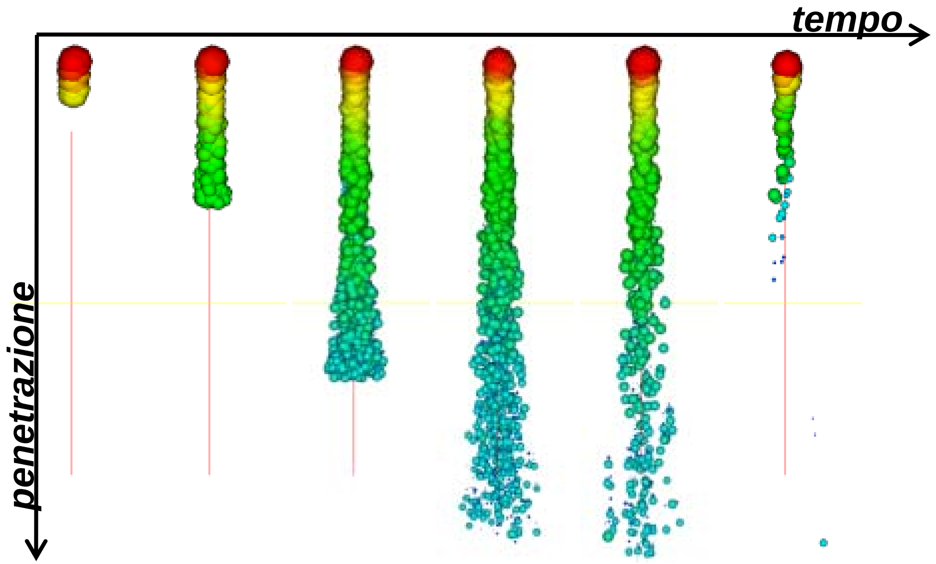
1. Combustion process

- Spray model calibration, throught experimental data of SANDIA database:

- Test in costant volume chambre, variable ambient conditions
- Typical common-rail injection simulation
- Misuration of spray penetration, ingnition delay, soot
- D2 fuel simulation (like diesel fuel)

- The results are validated on the base of liquid spray penetration

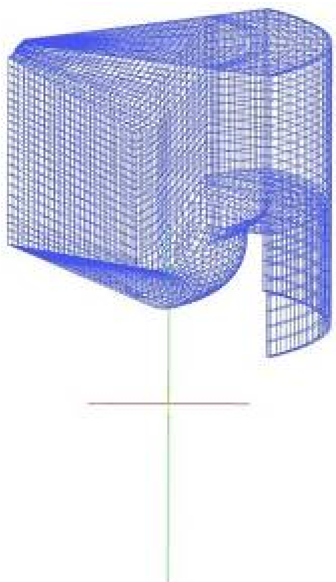




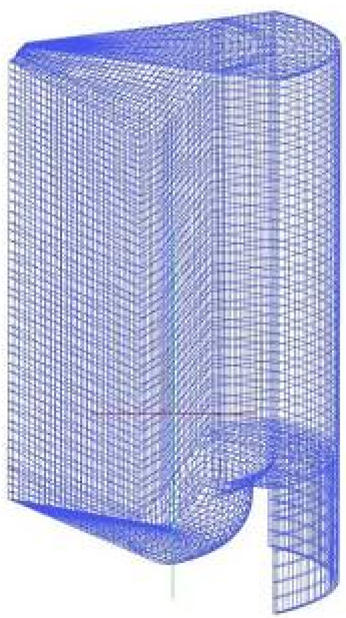
1. Combustion process

- 880 CR validation with experimental data:
  - We create a mesh motion grid

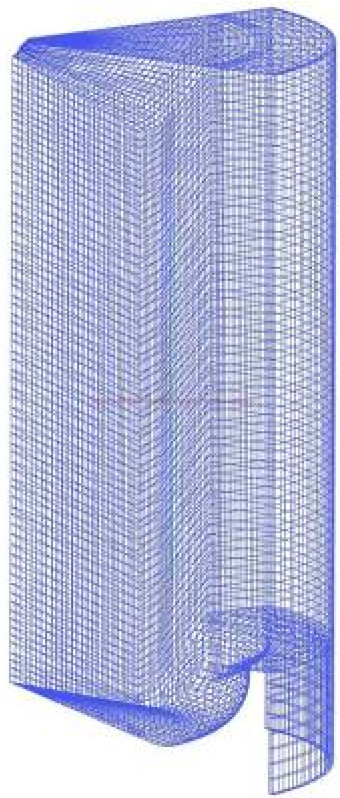
50° BTDC



90° BTDC

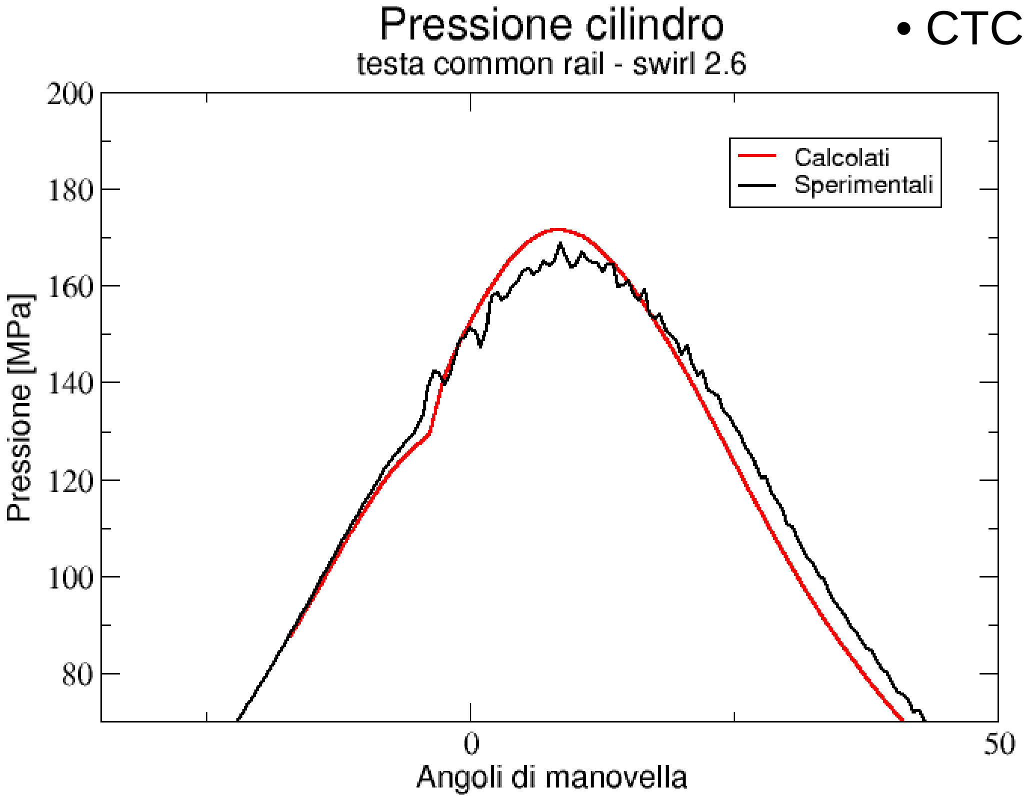


131° BTDC



1. Combustion process

- ➢ Configuration:
- • 3100 rpm, full load
- • Injection delay 22°
- • rail pression 150 Mpa
- • Combustion gemetry: 5 holes injector, swirl 2
- • CTC combustion model



1. Combustion process

1. Model validation
   - Spray validation
   - Engine data validation
2. Optimization of combustion process
   - Injection nozzle geometry
   - Effect of Swirl coefficient
   - Injection pressure

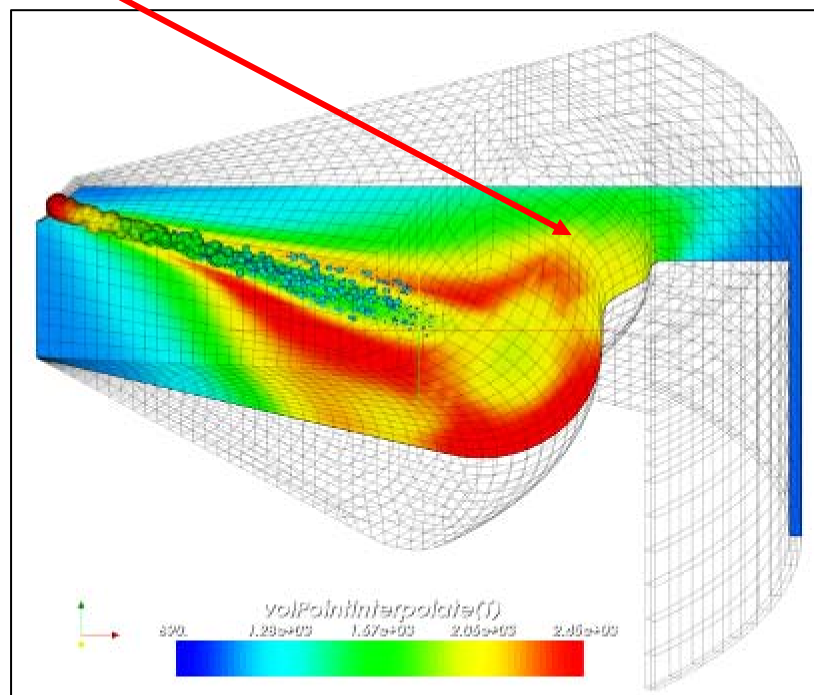
1. Combustion process

- Optimization of combustion process at max power output and study of the iterations between these factor

TARGET: limiting wall ITERATION

Img. 1

Img.2

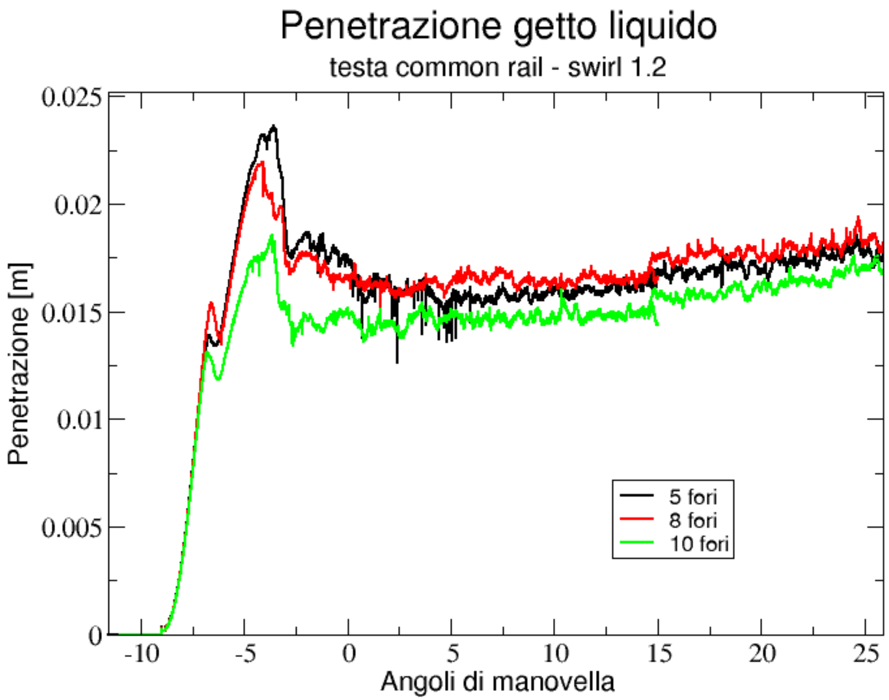


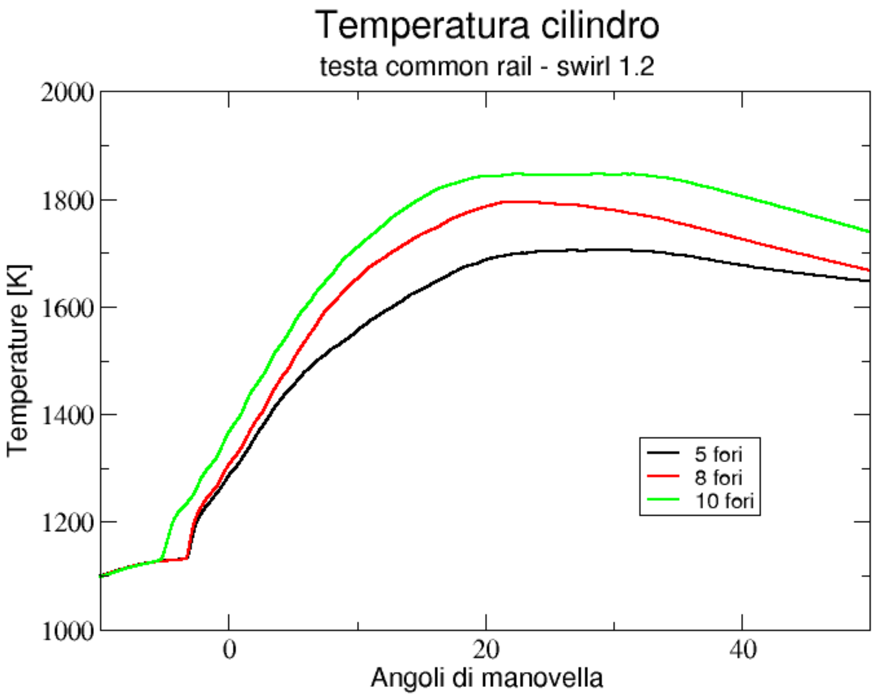
1. Combustion process

➢ Increase of nozzle holes number (current 5)

1. Less spray penetration
2. Better mixing phase
3. Less ingnition delay
4. Excessive increase of temperature (10 holes)

	0.8	1.2	2.4
5 fori	x	x	x
8 fori	x	x	x
10 fori	x	x	

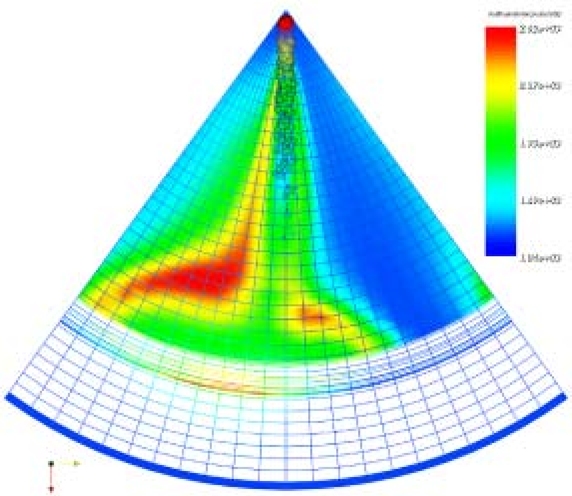




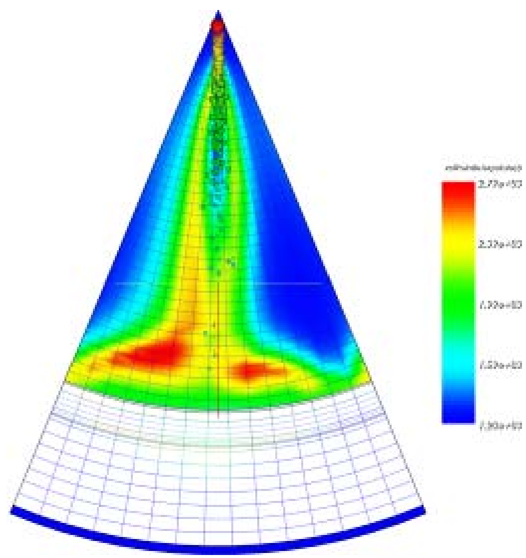
1. Combustion process

- Effects of raise nozzle holes number (current 5)

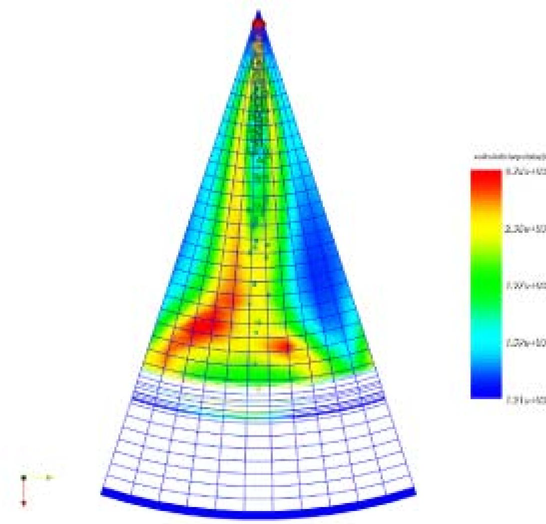
5 holes



8holes



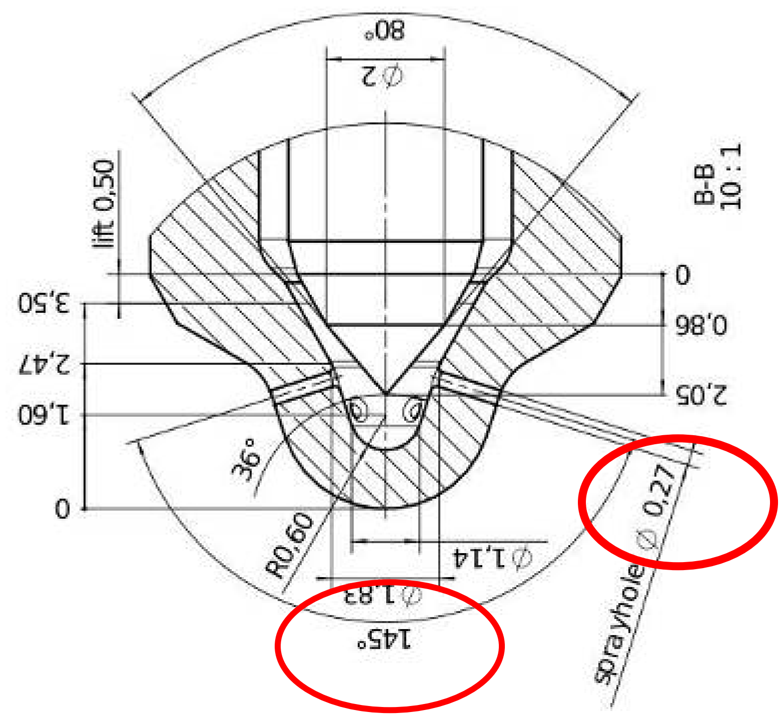
10holes



1. Combustion process

- Others fine nozzle optimization parameters

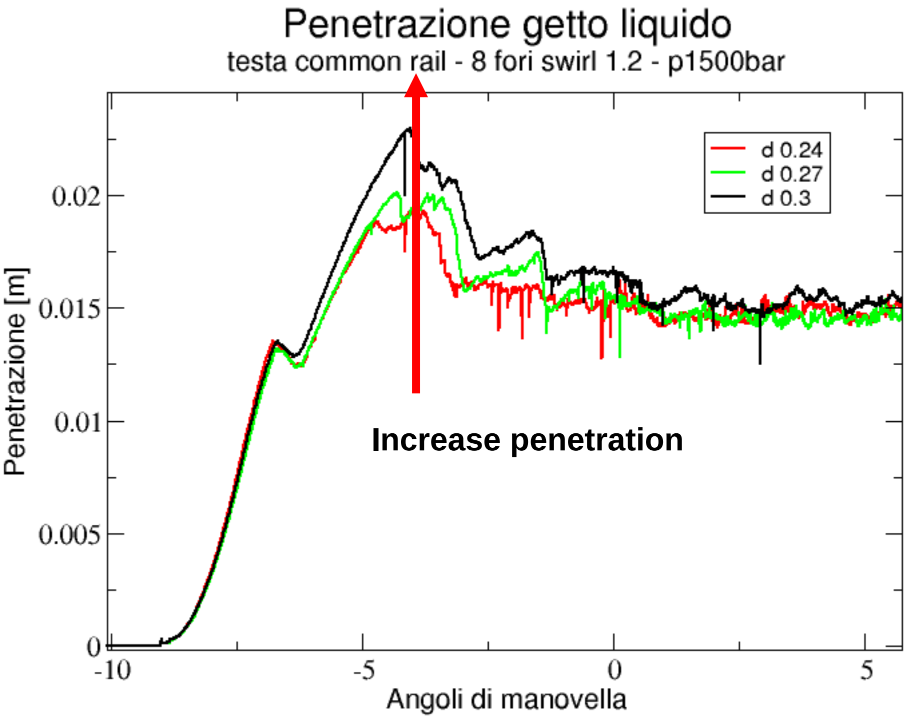
1. Diameter of holes
2. Spray Angle direction

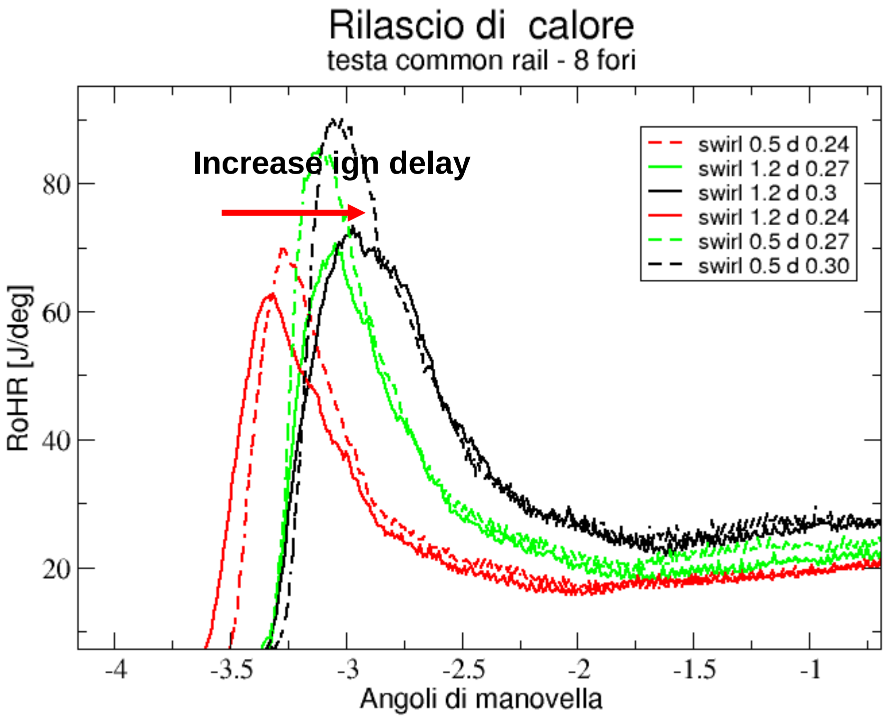


1. Combustion process

➢ Others fine nozzle optimization parameters

1. Diameter of holes
2. Angle direction of spray



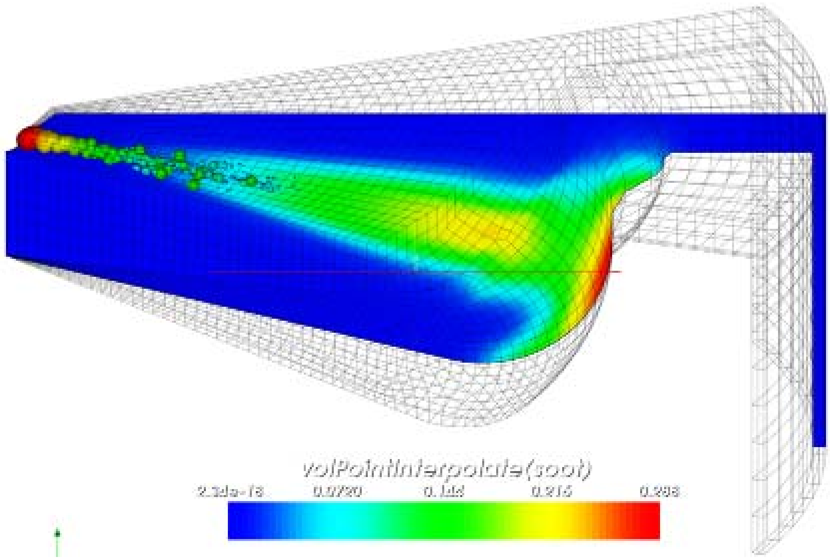


1. Combustion process

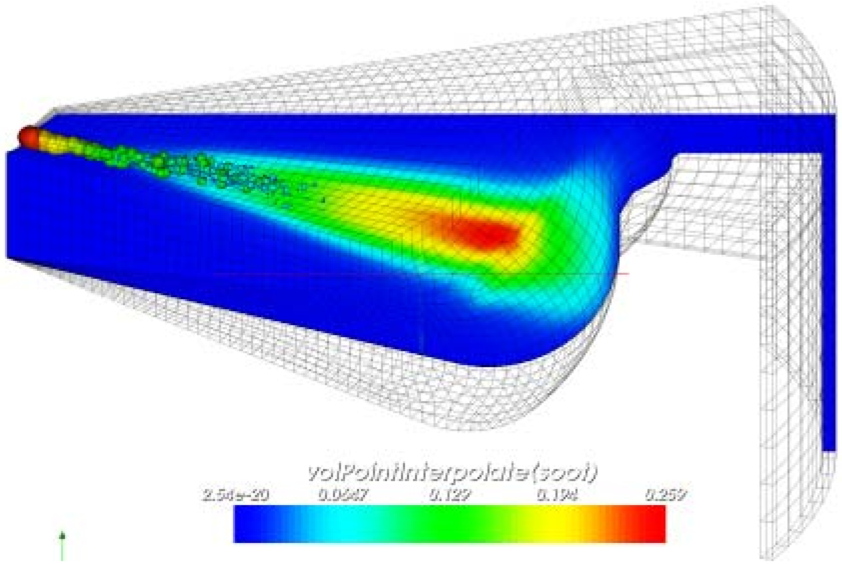
- Others fine nozzle optimization parameters

1. Diameter of holes
2. Angle direction of spray

0.27mm



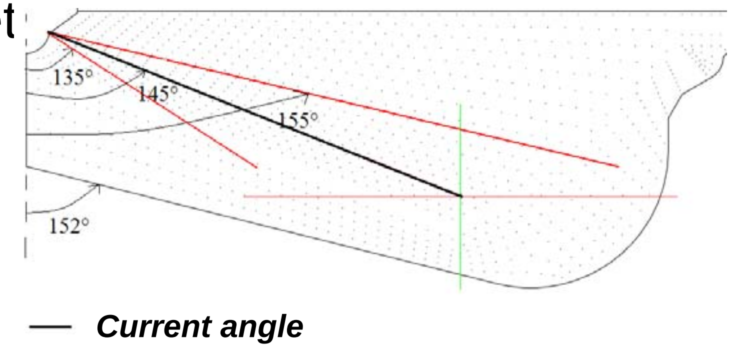
0.24mm



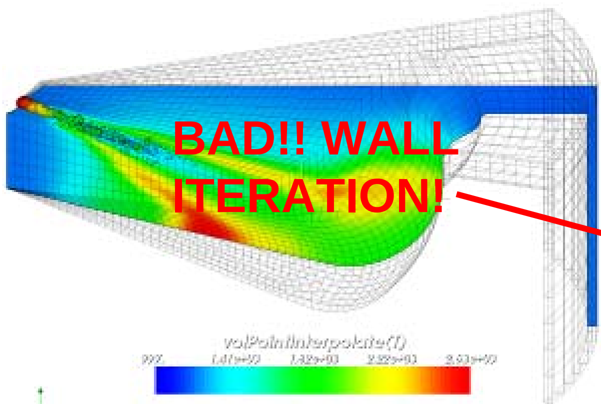
1. Combustion process

- Others fine nozzle optimization paramet

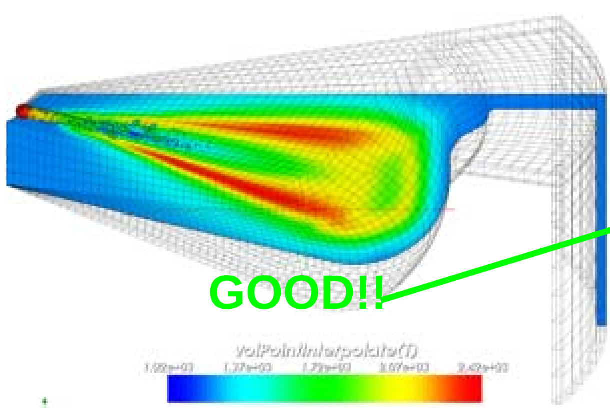
1. Diameter of holes
2. Angle direction of spray

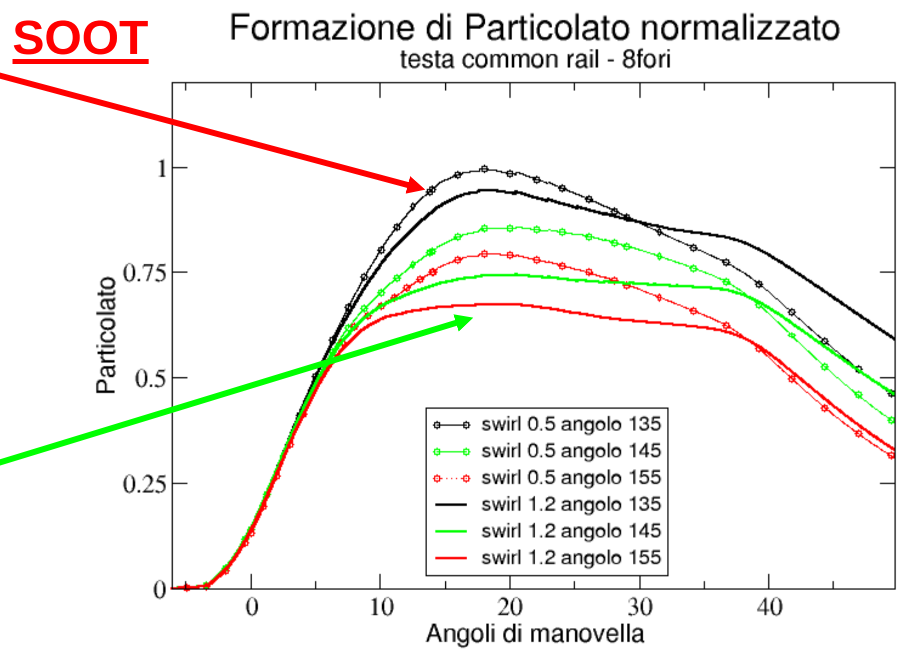


Angolo 135°



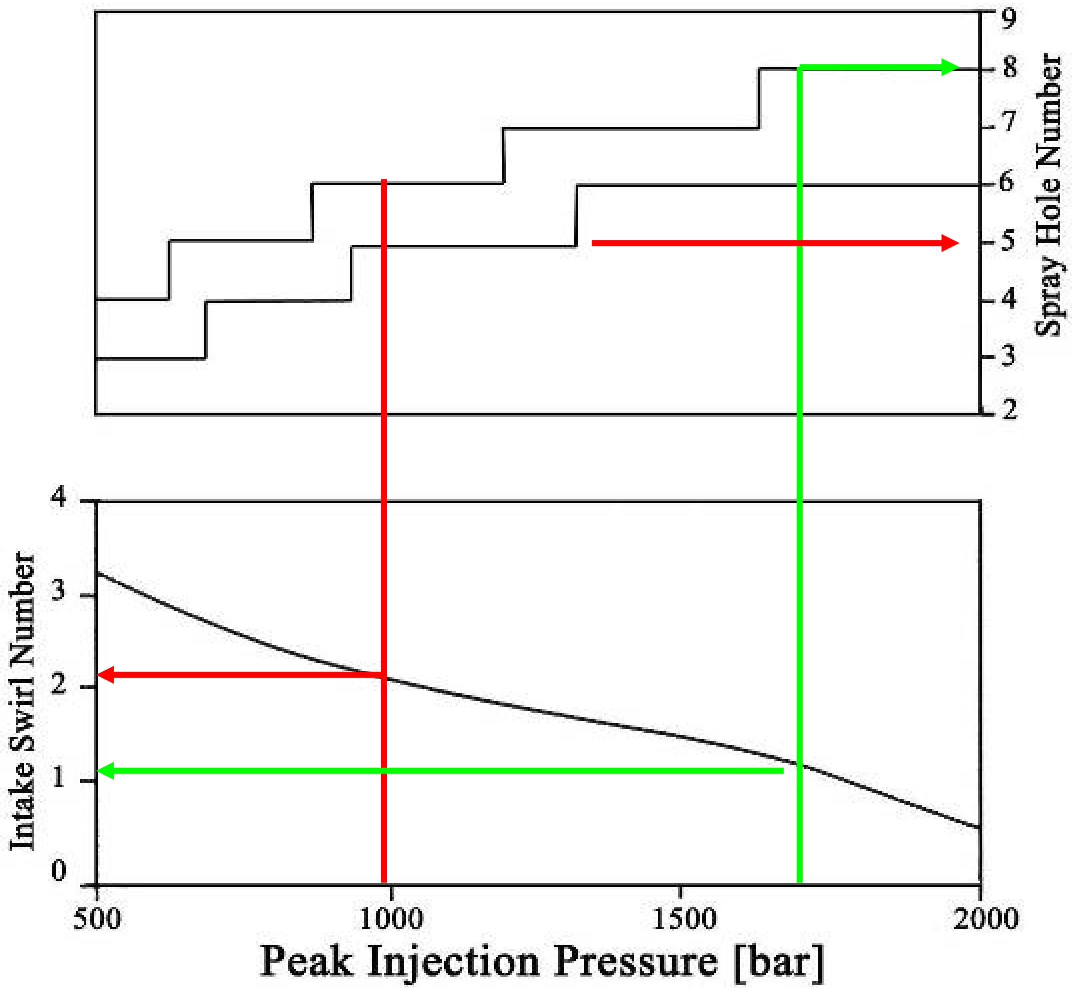
Angolo 155°





1. Combustion process

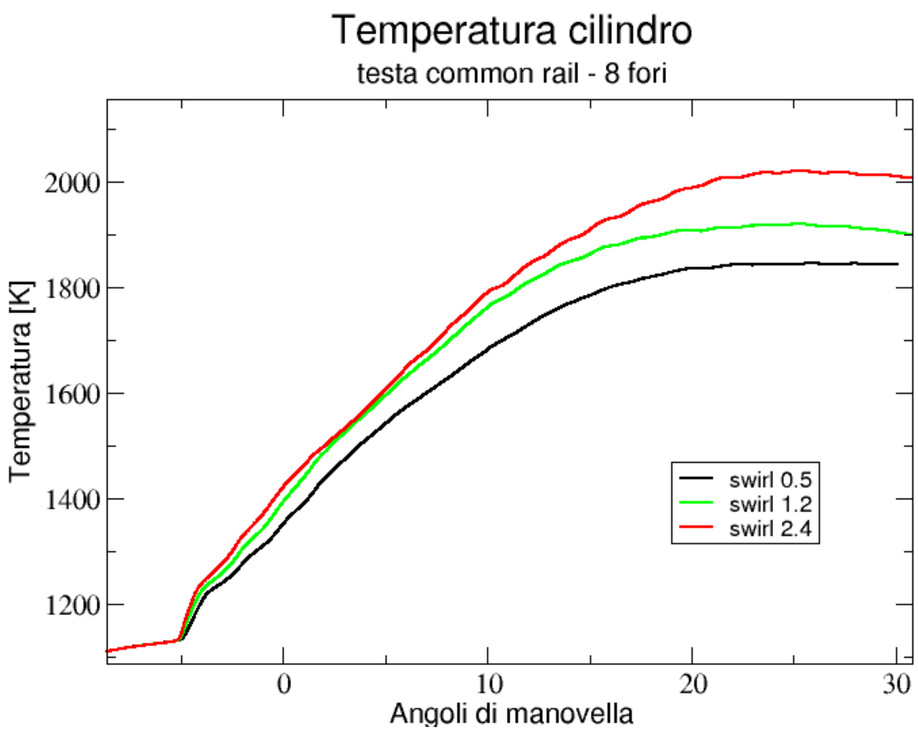
- Influence of swirl number on combustion process from AVL

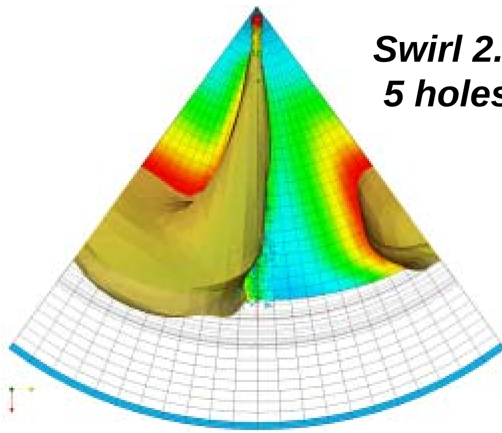


- With 8 holes decrease of Swirl we found:

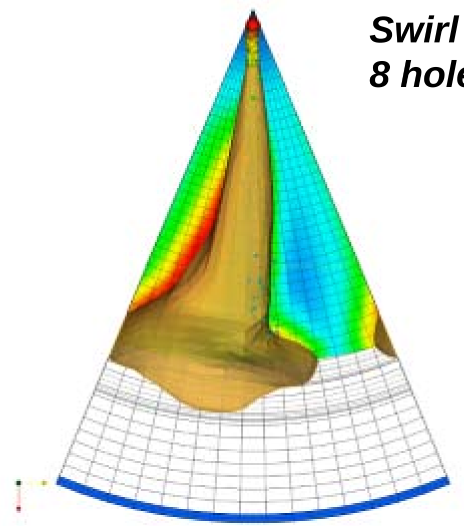
1. Decrease of maximun temperature
2. Decrease of heat peak release
3. Increase airflow on cylinder

$$N_{Swirl} = \frac{8M_s}{\dot{m}\,Dv_{is}}$$





Swirl 2.6
5 holes

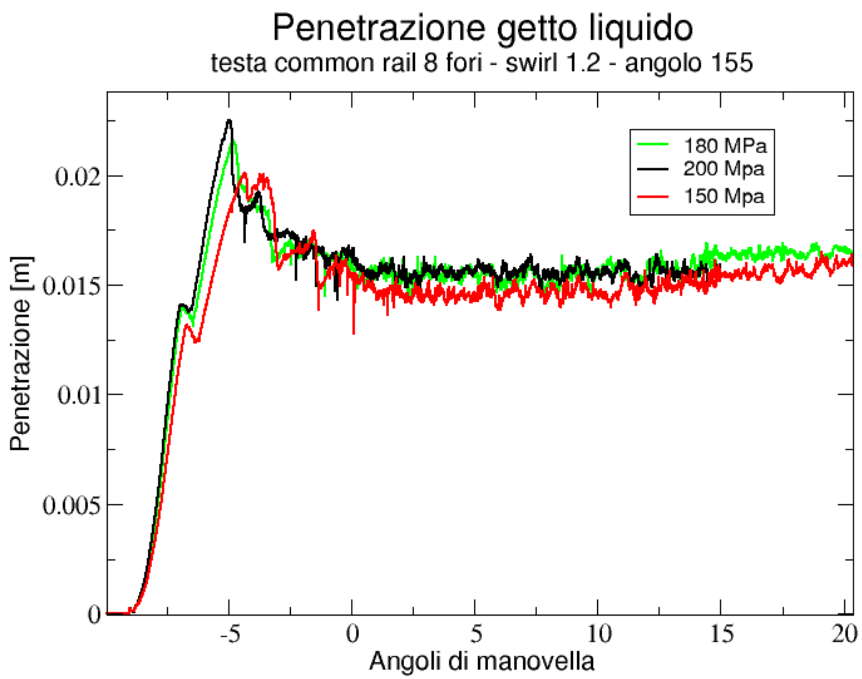


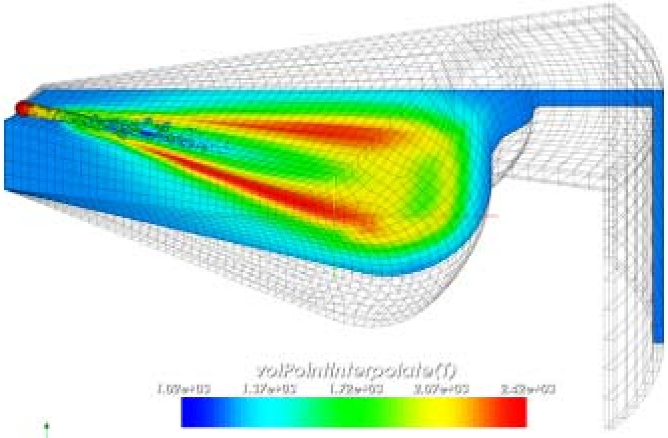
Swirl 1.2
8 holes

1. Combustion process

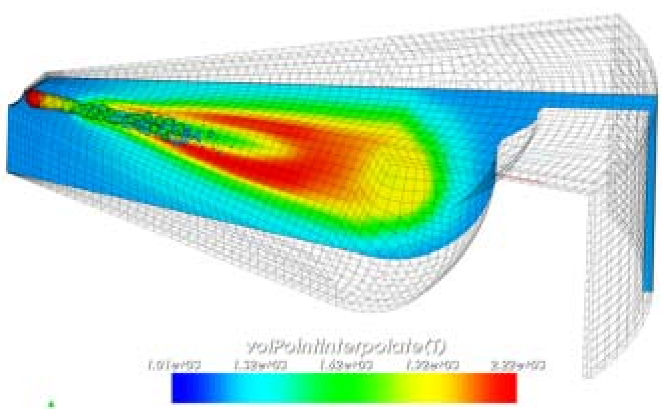
➢ Increase of rail pressure:

1. Better atomization of fuel
2. Increase cylinder pressure
3. Less injection time (same quantity inj.)
4. Increase of spray penetration





Rail pressure 150MPa



Rail pressure 200MPa

1. Combustion process

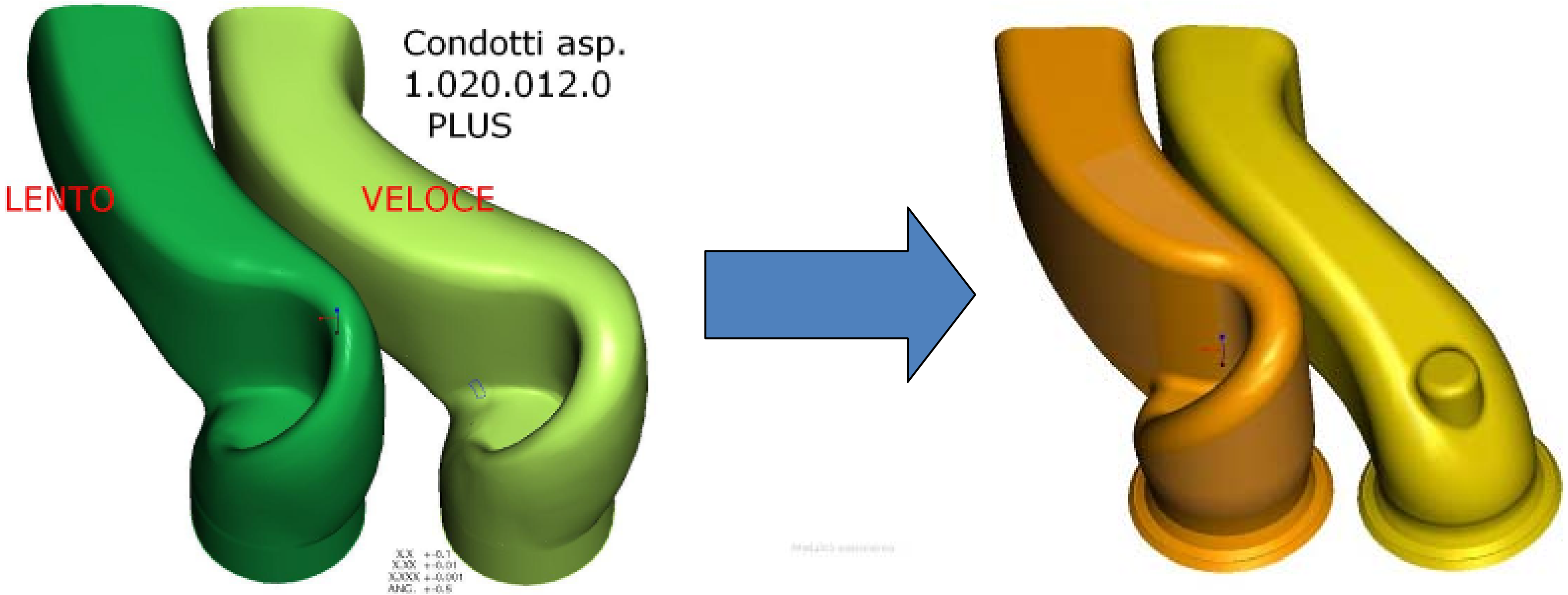
➢ The best optimization found for test bench development was:

	Trend	Value
Holes number	*Raise*	8
Swirl Number	*Reduction*	1.2
Hole diameter	*Reduction*	0.24 mm
Holes angle	*Raise*	155°
Rail pressure	*Raise*	200 MPa

➢ Further development

- Multi-injection
- Improved model calculation for emissions prediction

2. Cylinder head design of inlet ducts



2. Cylinder head design

- To achieve the result of combustion analisys is necessary a redesing of inlet duct of cylinder head in order to:

- Reduce swirl coefficient Rs from 2 to 1.2-1.5
- Improve discharge coefficient Cf
- Increase airflow value

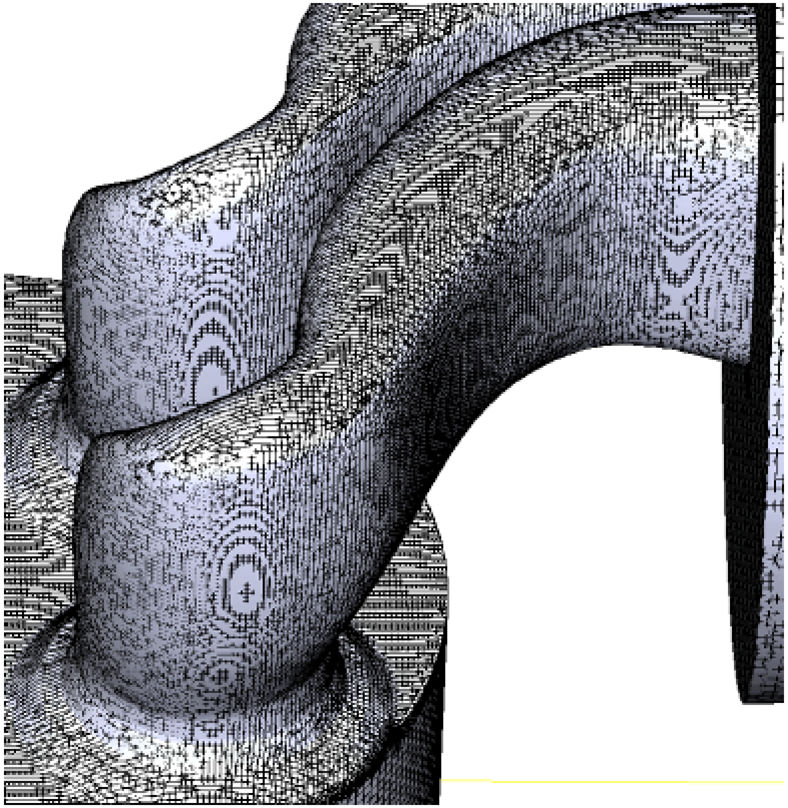
THIS REQUIRE…..

2. Cylinder head design

➢ Mesh features:

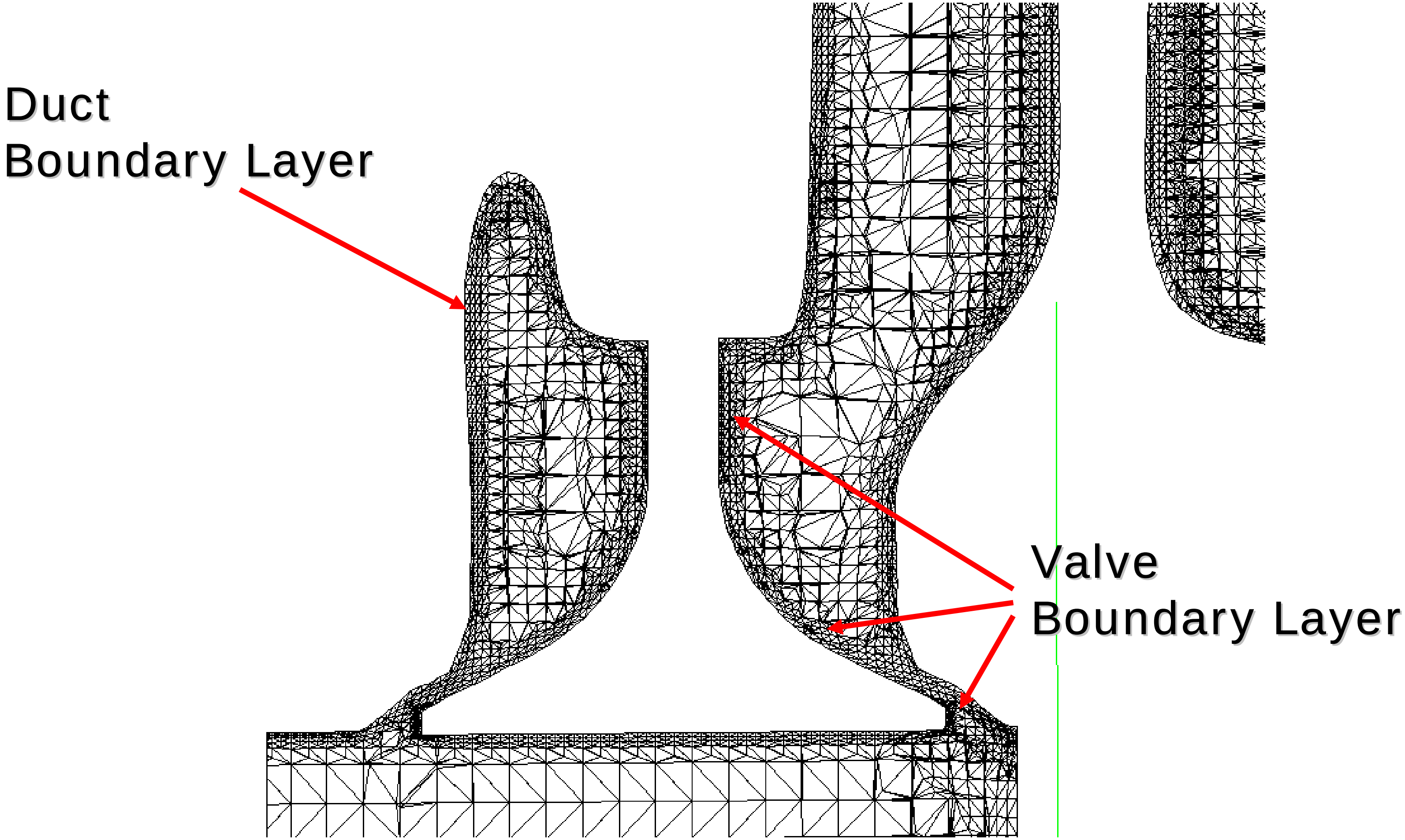
- Cartesian mesh generator
- Internal cell: max 3 mm;
- Duct boundary cell : 1 mm;
- Valve buondary cell : 1 mm;
- Lateral boundary cell : 1 mm;
- Head boundary cell : 1 mm
- Other boundary cell: 2mm;

- TOTAL: 900000 cells



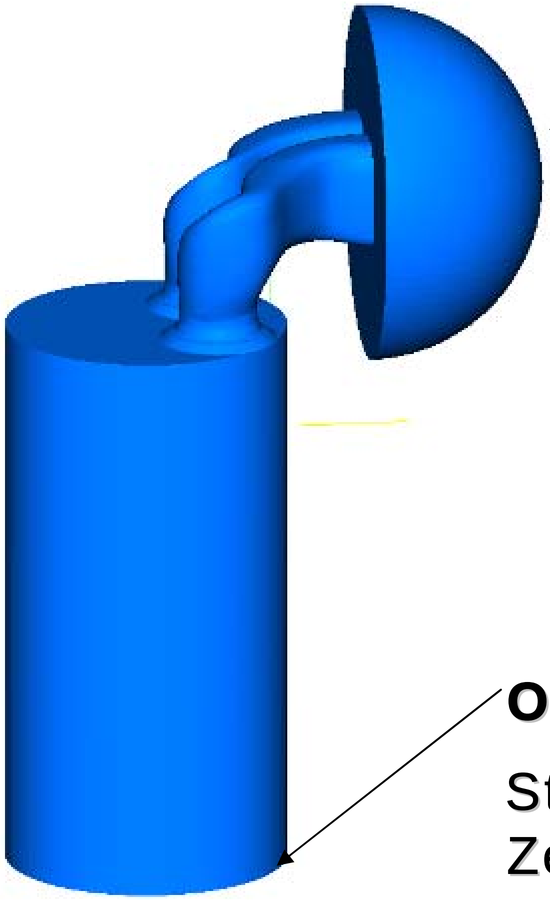
2. Cylinder head design

- Mesh features:



2. Cylinder head design

- Boundary conditions



INLET

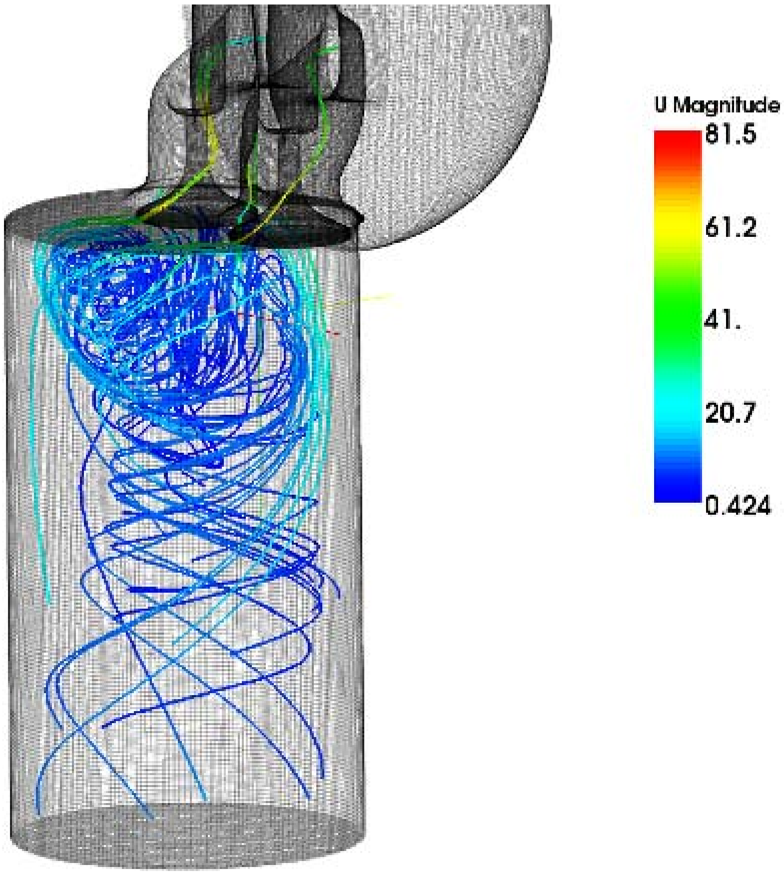
Total pressure = 0.978 bar
Temperatura = 303 K
$k = 1\ m^2/s^2$
$\varepsilon = 90\ m^2/s^3$

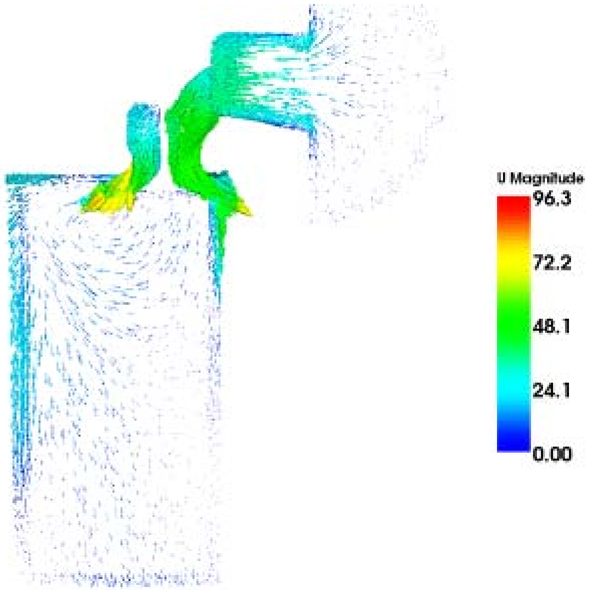
OUTLET

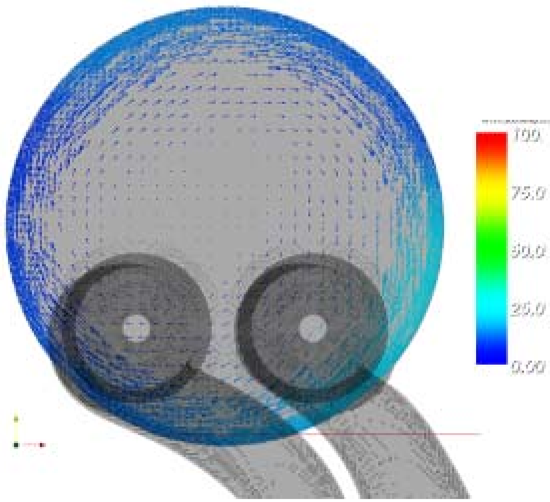
Static pressure = 0.954 bar
Zero gradient for others parameters

2. Cylinder head design

- Flow field examples:

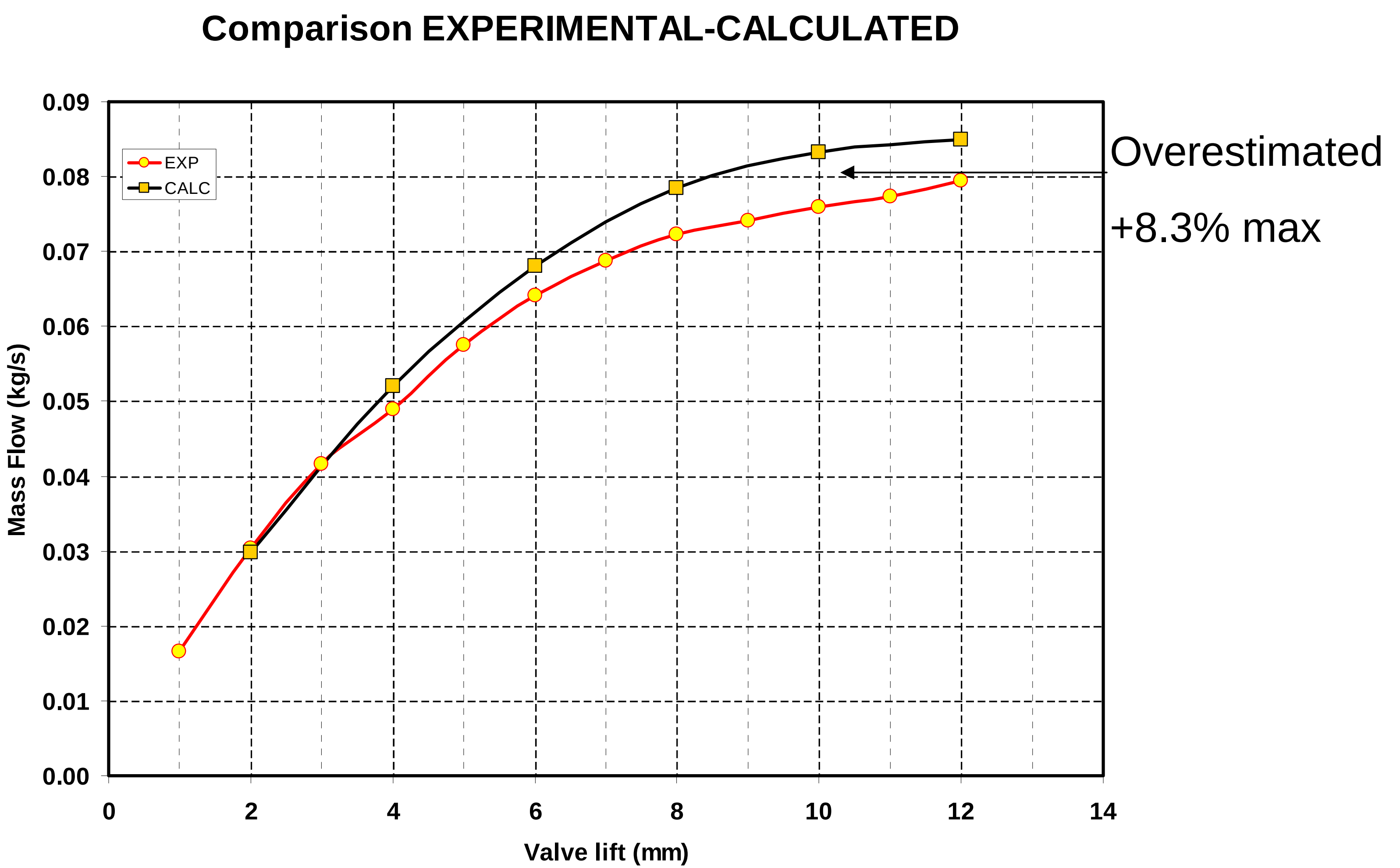






2. Cylinder head design

- Cf prediction



2. Cylinder head design

- Rs Swirl (Ricardo method) prediction:

$$Rs = LD*\left(\int C(\alpha)*Nr(\alpha)\right)/\left(\int C(\alpha)d(\alpha)\right)^2$$

- Swirl torque isn't calculated because the simulations are do at stationary condition, for that reason we normalized the angular momentum quantity on the maximun value at max valve lift

$$C_{s,exp} = \frac{T_{swirl}}{T_{swirl,h=12mm}} \qquad C_{s,calc} = \frac{M_z}{M_{z,h=12mm}}$$

2. Cylinder head design

- Rs Swirl (Ricardo method) prediction:

- Moreover we observe, ***only for comparison***, that there's a good accordance between the calculated swirl on test bench:

$$Rs = LD * \frac{\sum_{I=1}^{h} Cf(\alpha) * Nr(\alpha)}{\left(\sum_{i=1}^{h} Cf(\alpha) * \alpha\right)^2}$$

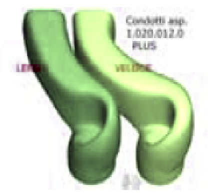
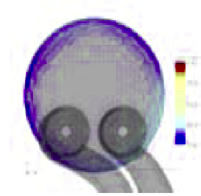
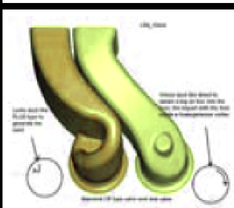
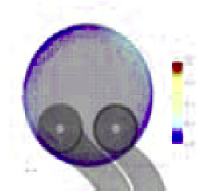
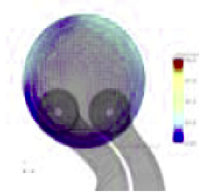
And the summatory of normalized angular momentum quantity:

$$RS_{calc} = \sum_{i=1}^{h} Cs_{calc}(h) * \alpha(h)$$

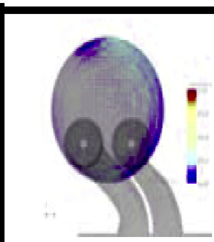
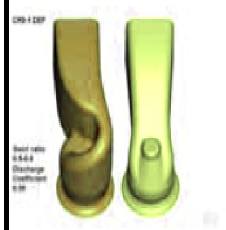
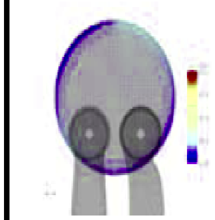
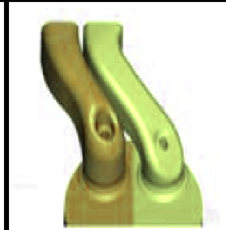
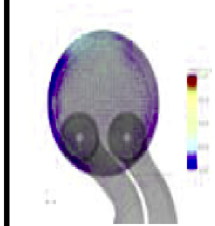
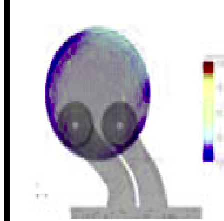
2. Cylinder head design

➢ Good approximation with result found on test bench

Simulazione condotti CR 2008/2009

SWIRL Rs		NOTE	FOTO	CAMPO	NOTE	Cf	Rs	Cf	Rs
	Tipo					Simulati		Sperimentali	
1.5-2	PLUS	Condotti originali di partenza			Vortice ben definito e centrato	0.33	1.79	0.33	1.90
1.2-1.5	CR6-3	Modello simulato e realizzato gennaio 2008			Vortice ben definito e centrato	0.36	1.56	0.36	1.20
	CR6-33				Vortice ben definito e centrato	0.37	1.49		

SEATEK ADVANCED MARINE PROPULSION TECHNOLOGY

0.5-0.8	CR6-35				Vortice definito e centrato dall'alzata 8mm	0.36	0.72		
	CR9-1	Modello simulato e realizzato ottobre 2008			Vortice definito dall'alzata 8mm ma non centrato	0.37	0.64	0.39	0.69
	CR6-31	Deciso realizzazione modello e testata a febbraio 2009			Vortice quasi definito dall'alzata 8mm ma non centrato	0.38	0.69	0.37	0.31
	CR6-37	Deciso realizzazione testata a Aprile 2009			Vortice definito e centrato dall'alzata 8mm	0.38	0.68		

2. Cylinder head design

- Advantages of “Virtual FLOW BENCH”

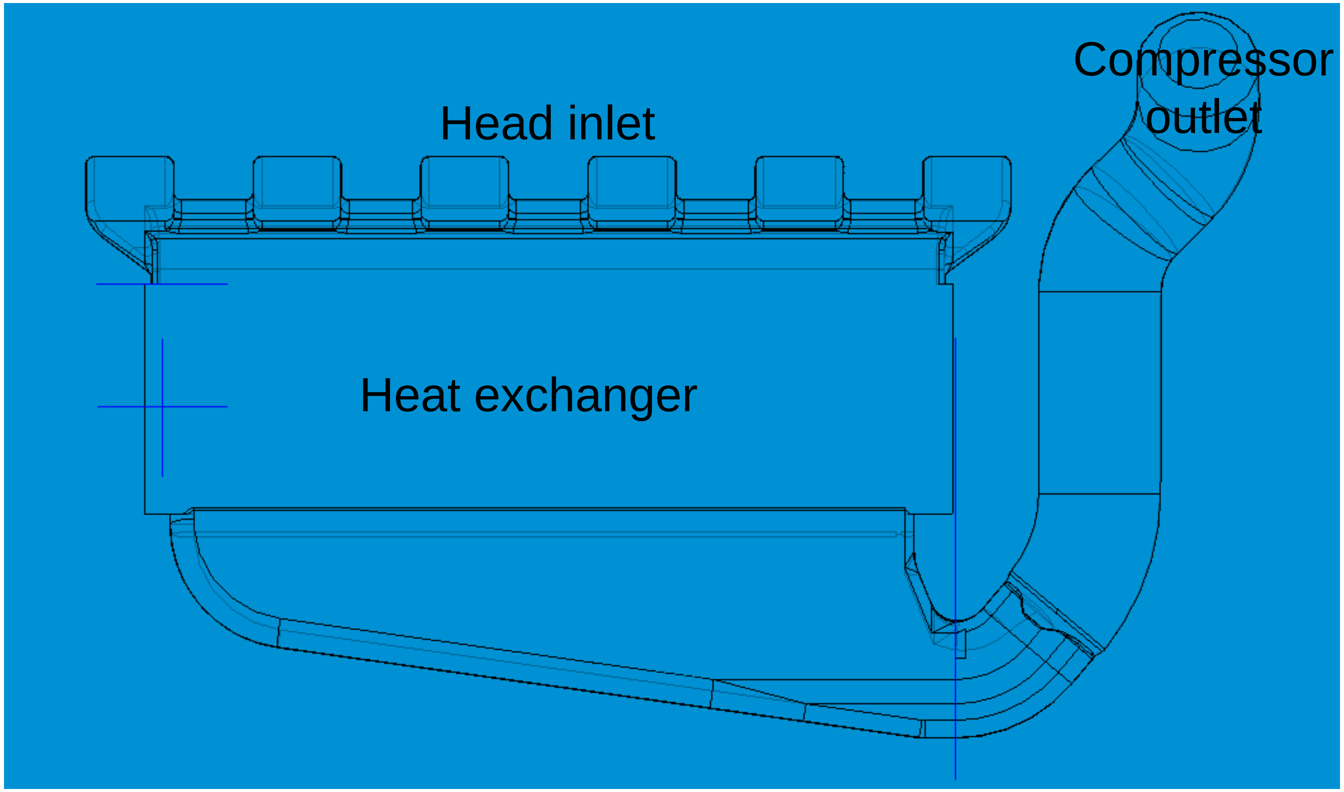
- Study of a large amount of type of ducts
- Good prediction of Rs and Cf coefficients
- It is possible evaluate the “shape” of airflowes and vortex
- Effective time and cost savings….

2. Cylinder head design

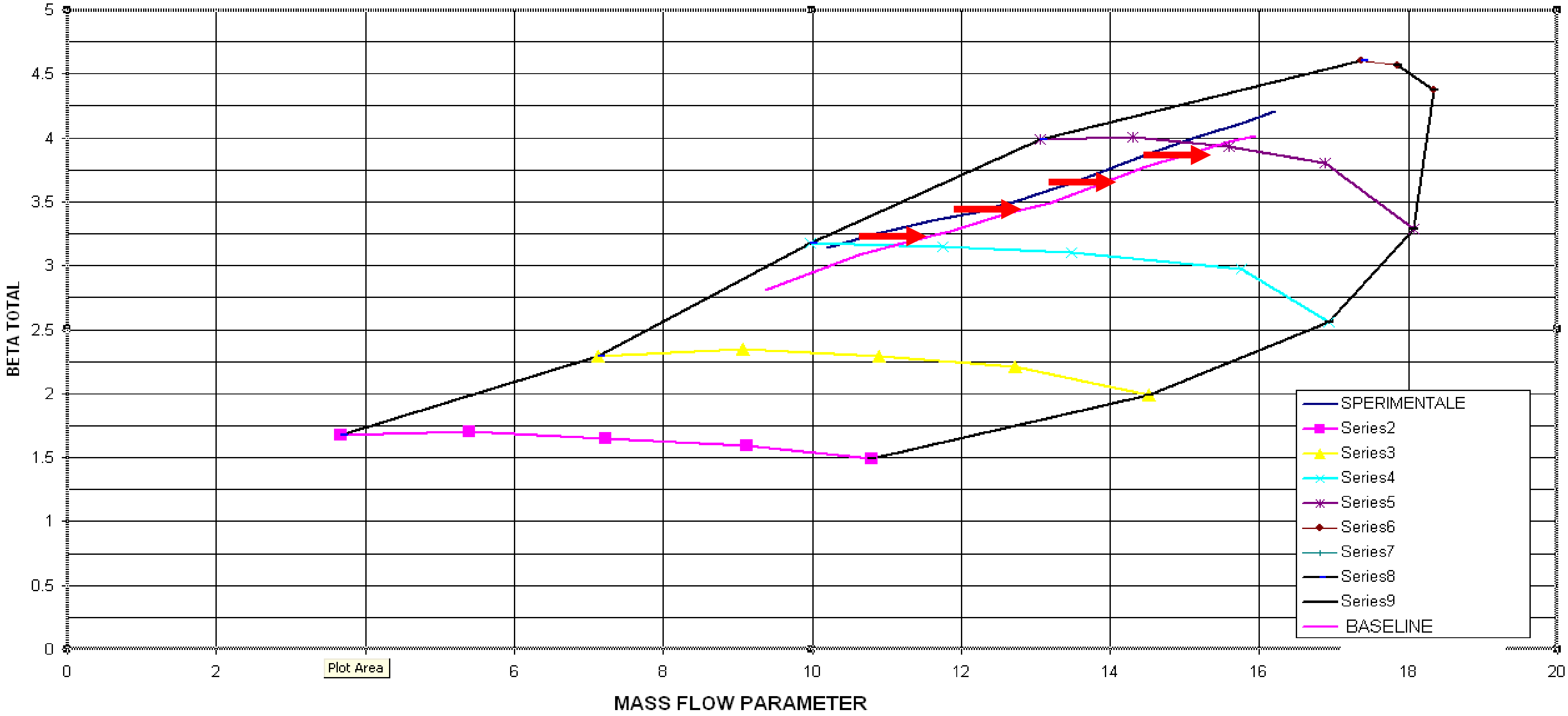
➢ Advantages of “Virtual FLOW BENCH”

….In 8 months we do analisys and realization of the first cylinder head for preliminary testing on common rail configuration

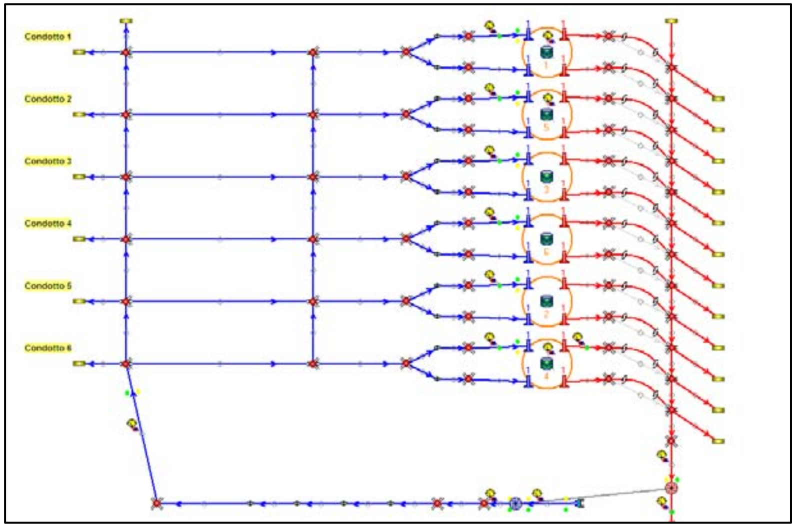
3. Intercooler housing design

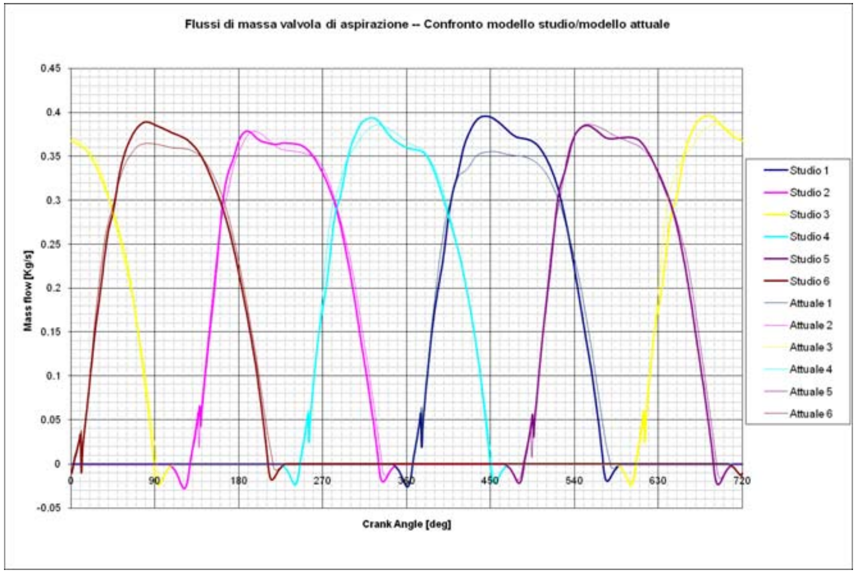


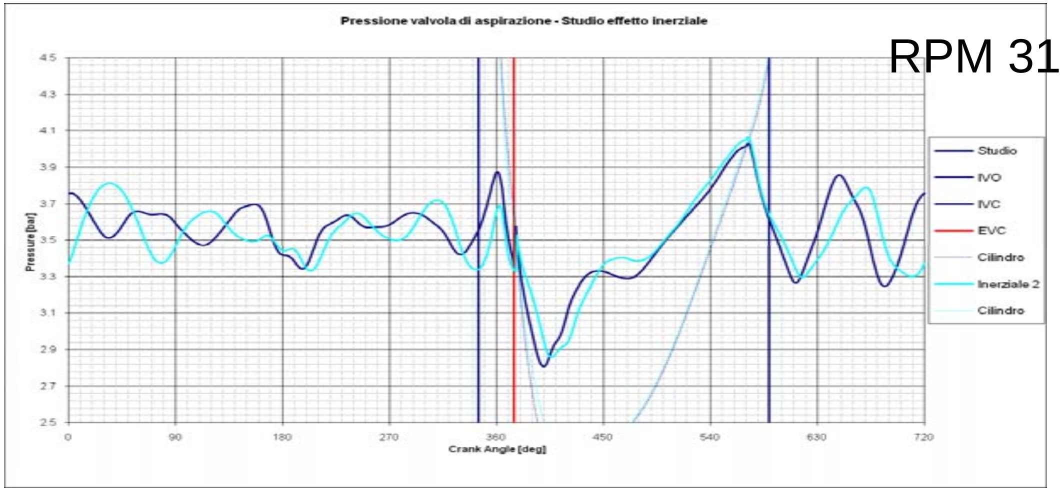
- Icrease the airflow value into the engine from TURBO



- 1D data from gasdyn simulation.



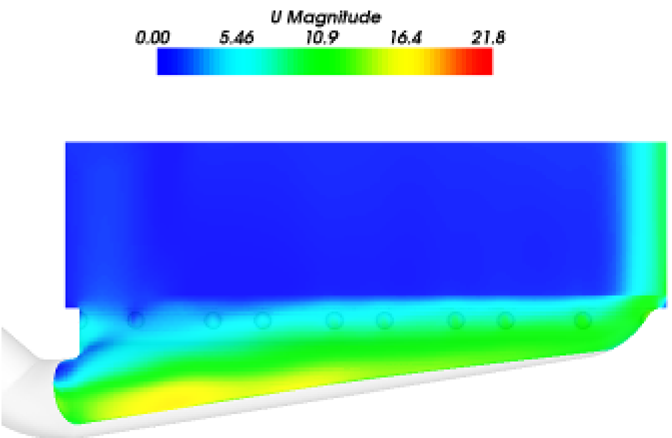


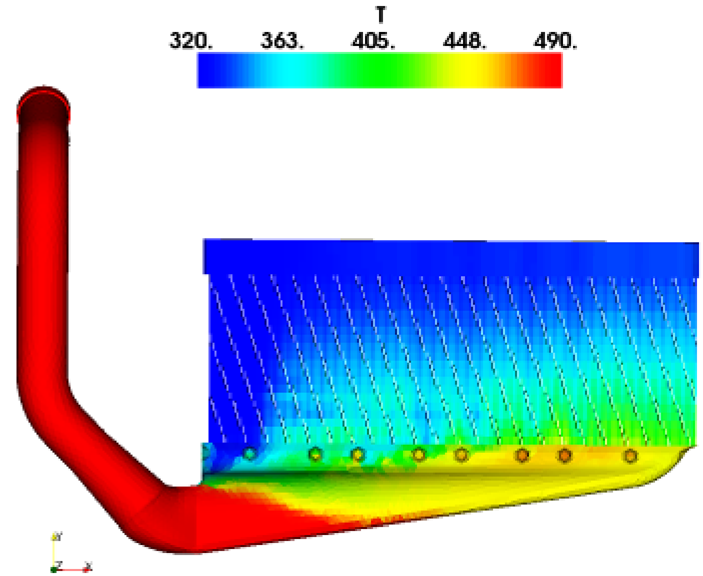


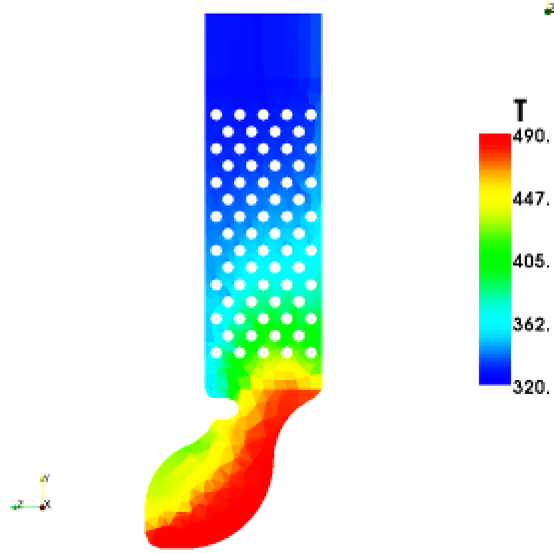
RPM 3100

3. Intercooler design

- Flow and temperature fields

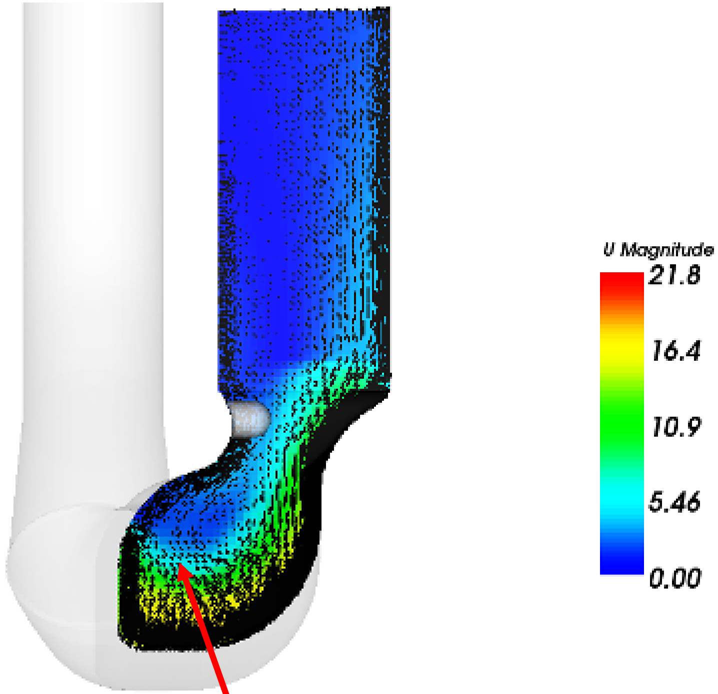




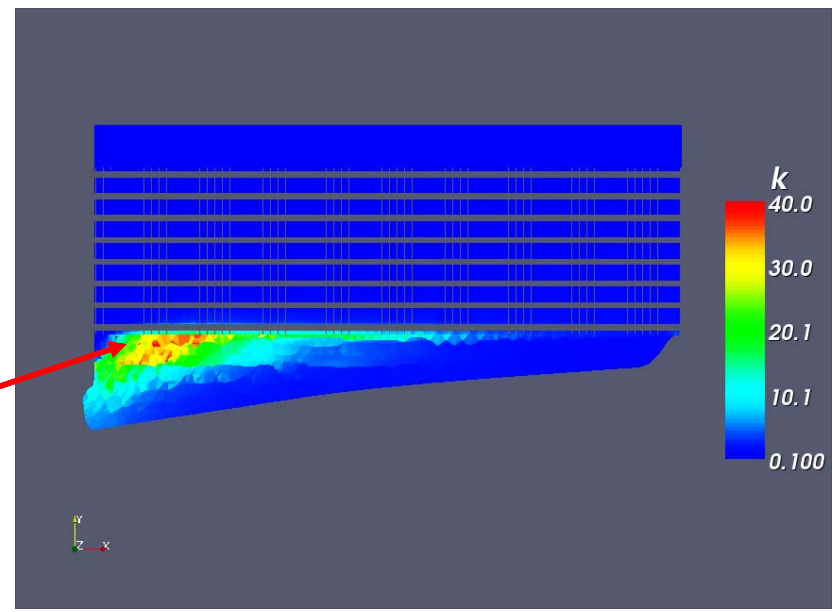


3. Intercooler design

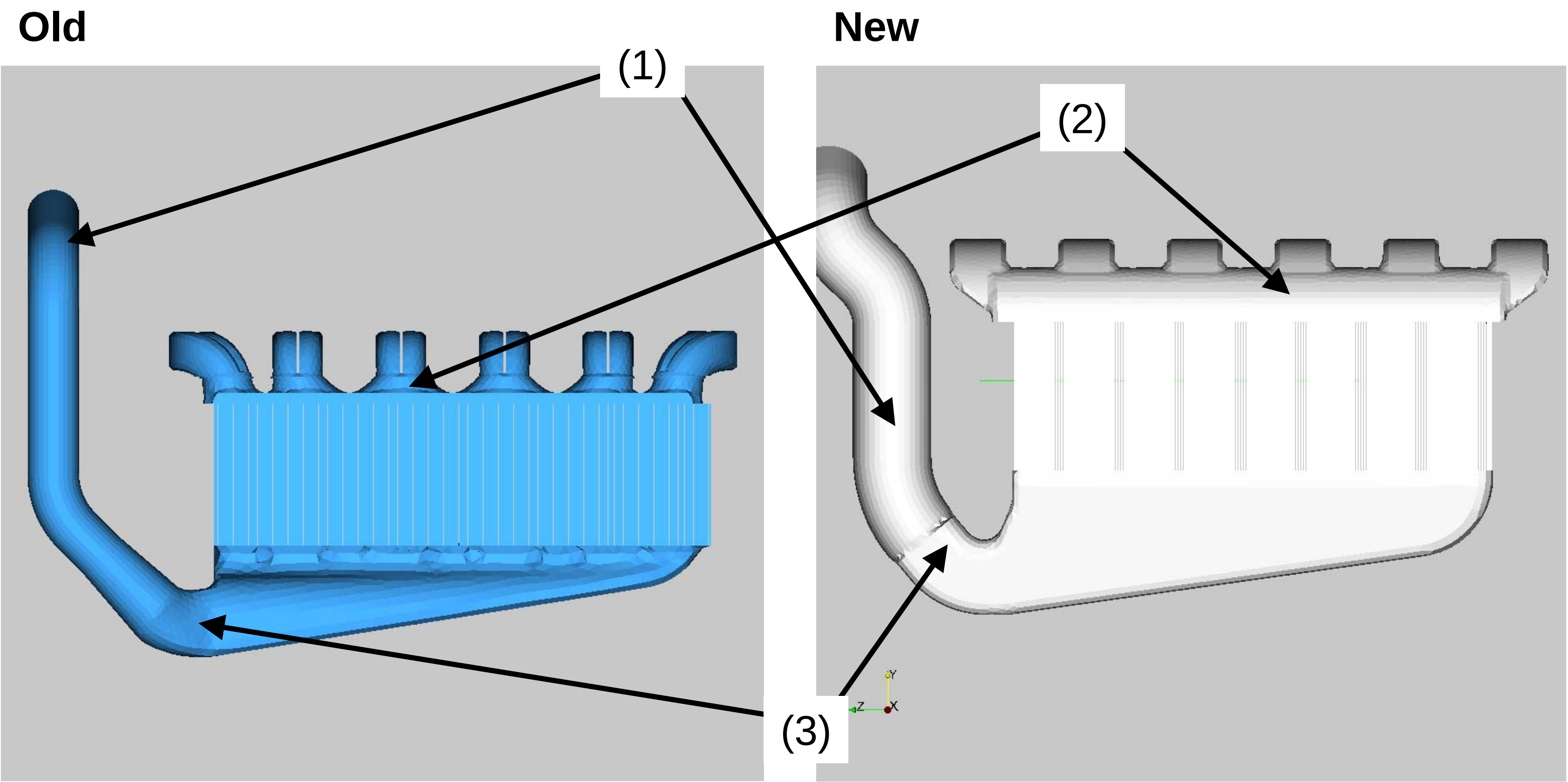
- Flow and temperature fields



Misalignement
create turbolence



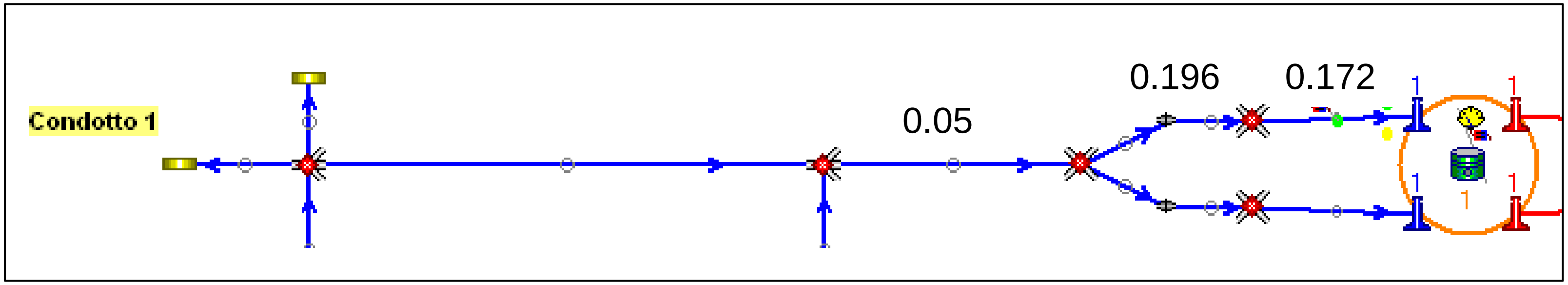
3. Intercooler design



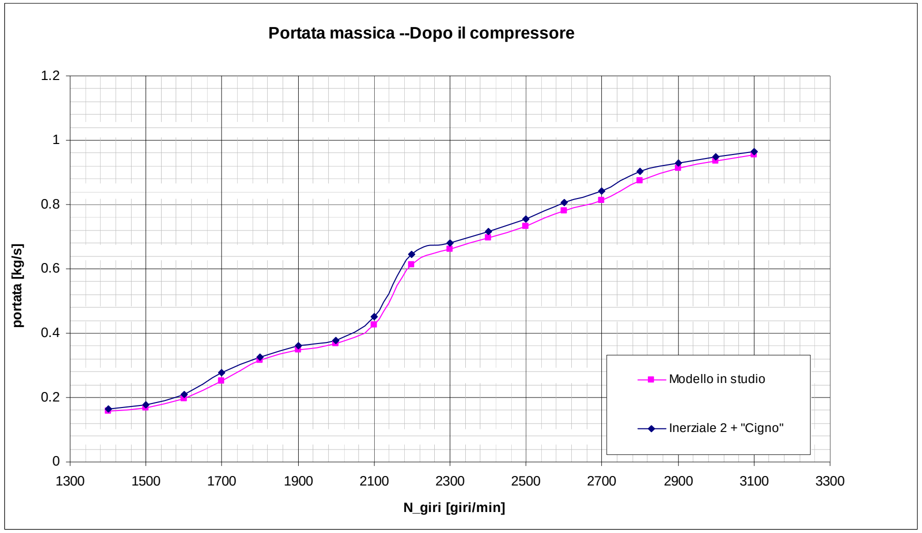
- Raise of inlet volume to cooler (1)
- Increase of upper volume of plenum (2)
- Improved shape of inlet in lower volume of plenum (3)

3. Intercooler design

➢ Improved geometry 1-D analysis



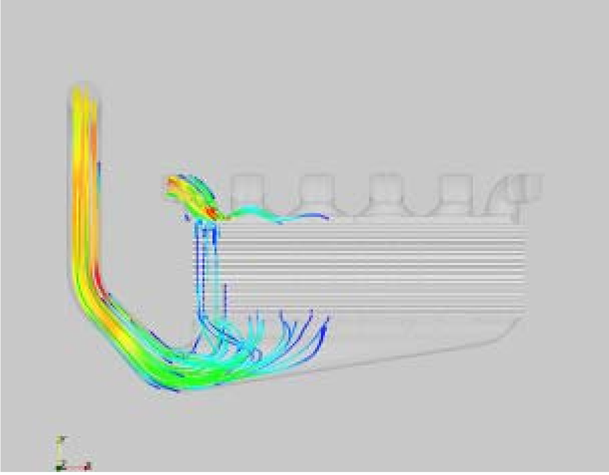
- Raise of inlet volume to cooler (1)
- Increase of upper volume of plenum (2)
- Improved shape of inlet in lower volume of plenum (3)

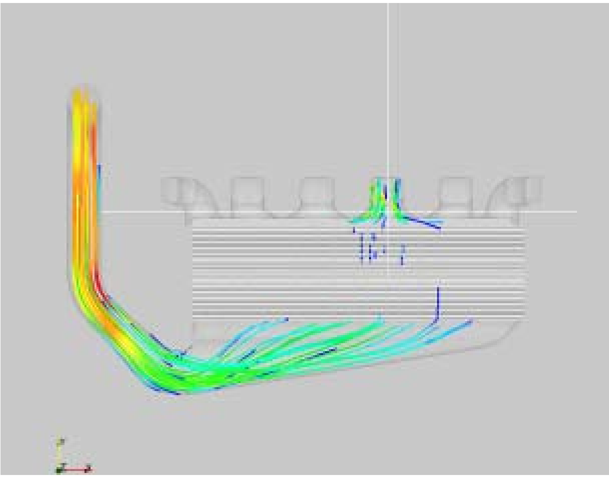


3. Intercooler design

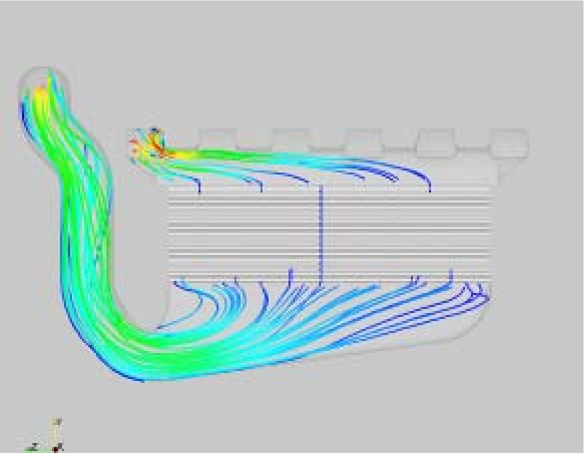
- Improved geometry

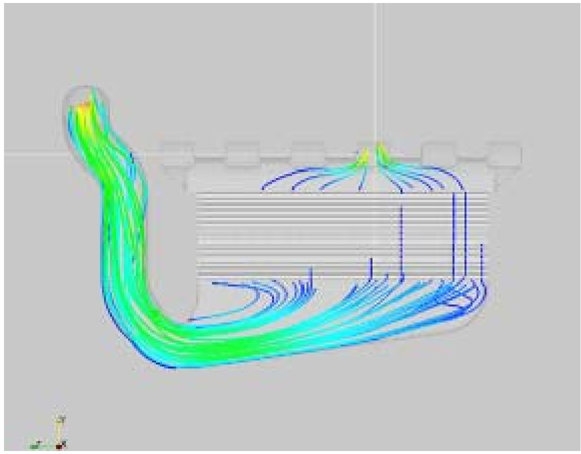
OLD



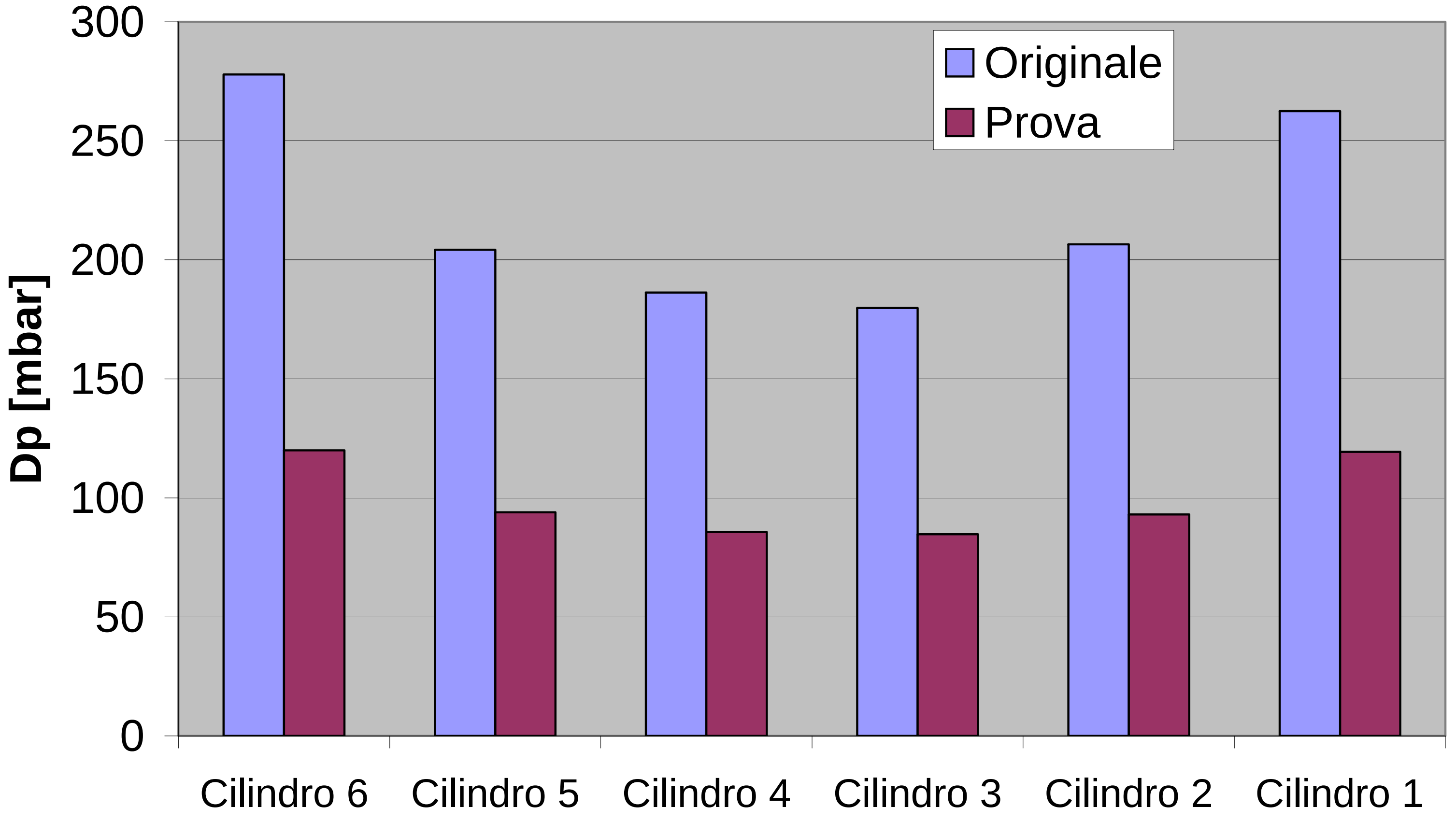


NEW

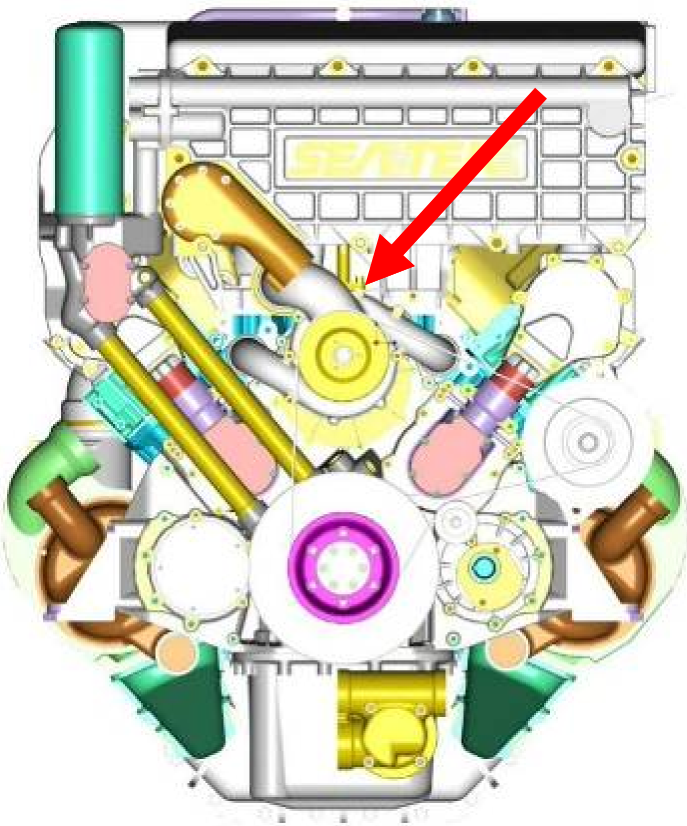


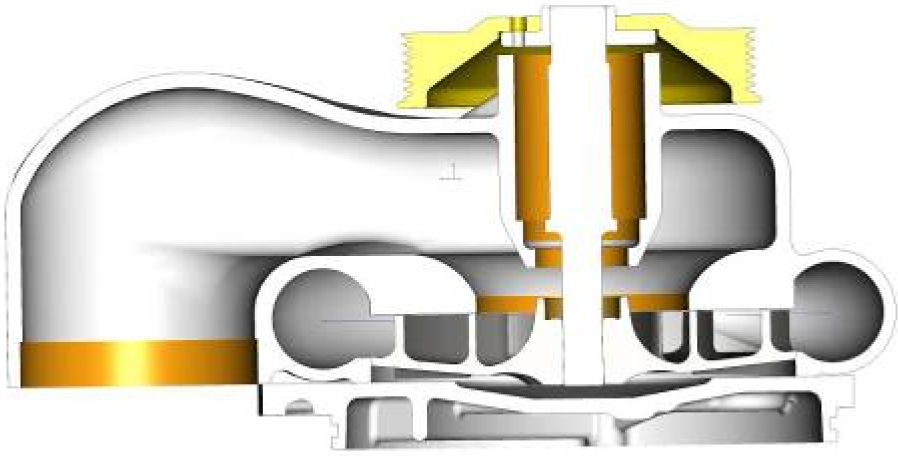


Variazione di pressione totale: uscita compressore - uscita intercooler



4. Water pump analysis



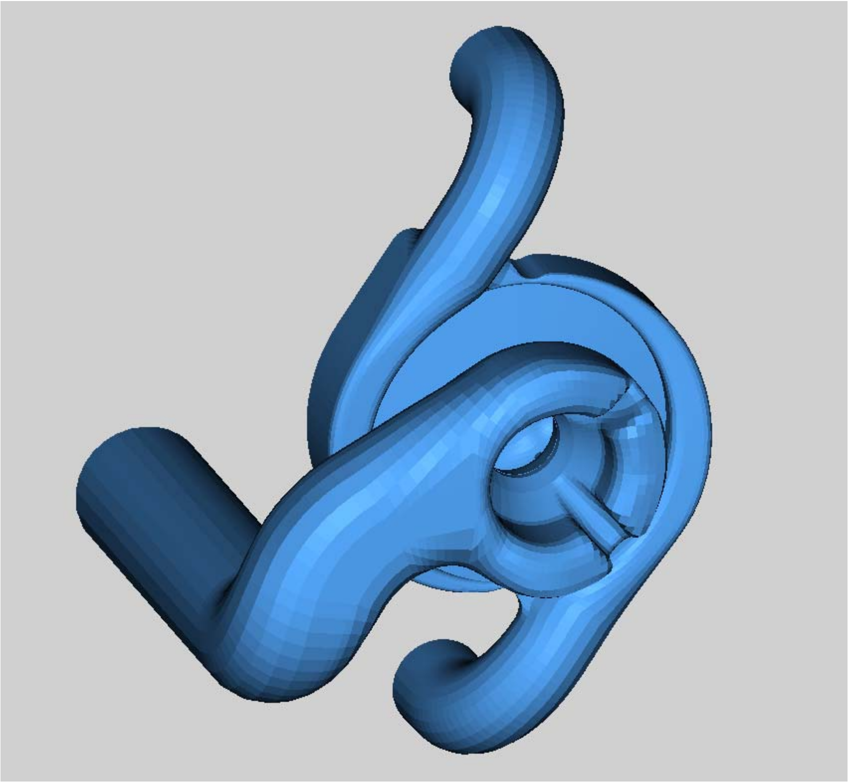


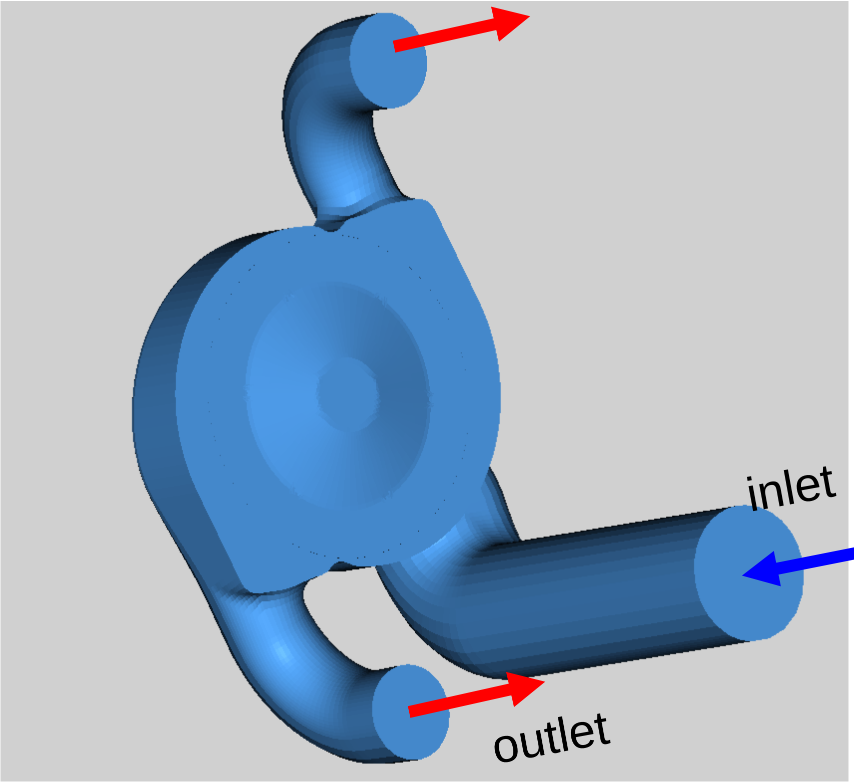
4. Water pump analysis

- Simulation features:
  - ✷ Rotation of the impeller
  - ✷ Turbolent and non-stationary flow with geometry movement.
  - ✷ Complex geometry
  - ✷ High calculation time for convergence
  - ✷ Manual creation of the mesh for calculation
  - ✷ GGI (generalized grid interface) approach for simulation of flow through the impeller

4. Water pump analysis

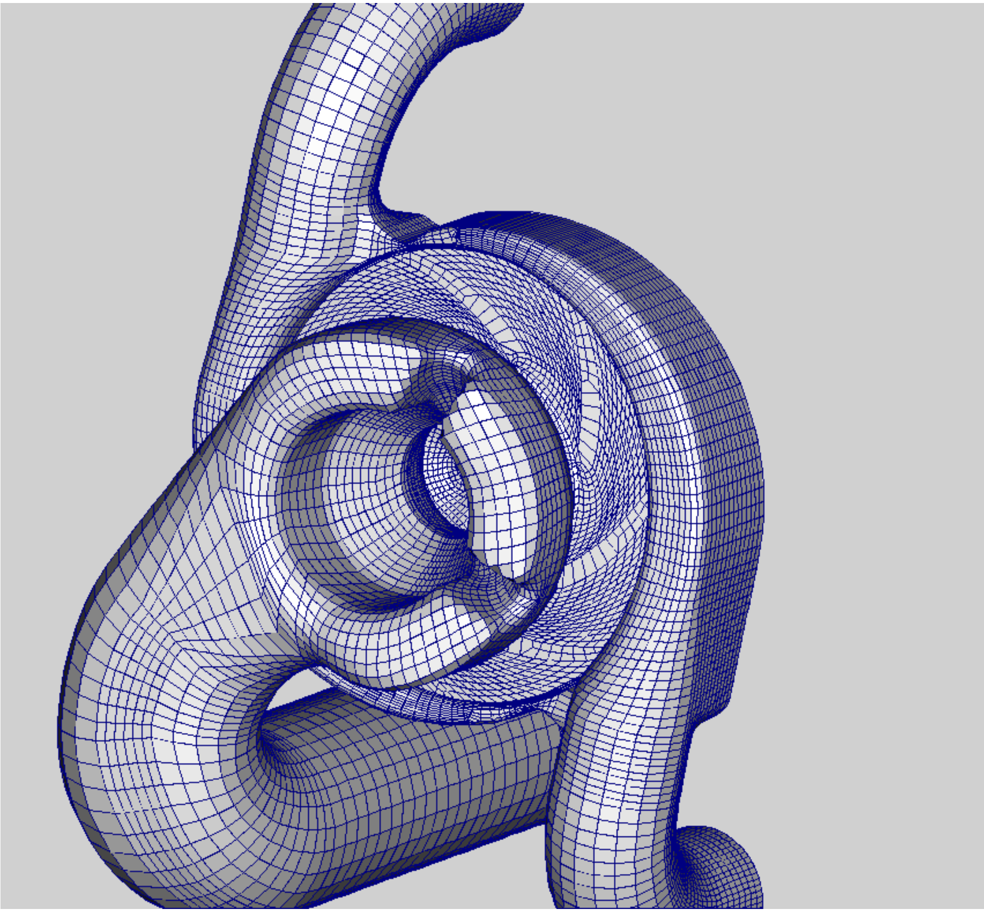
- The geometry include inlet and outlet ducts (it's for a VEE engine).



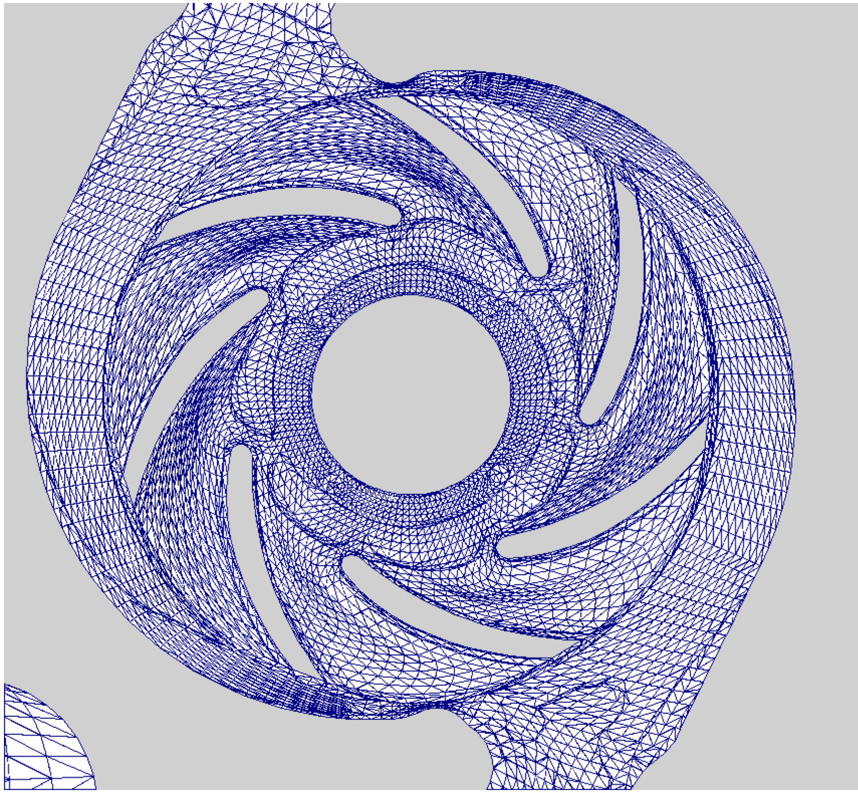


4. Water pump analysis

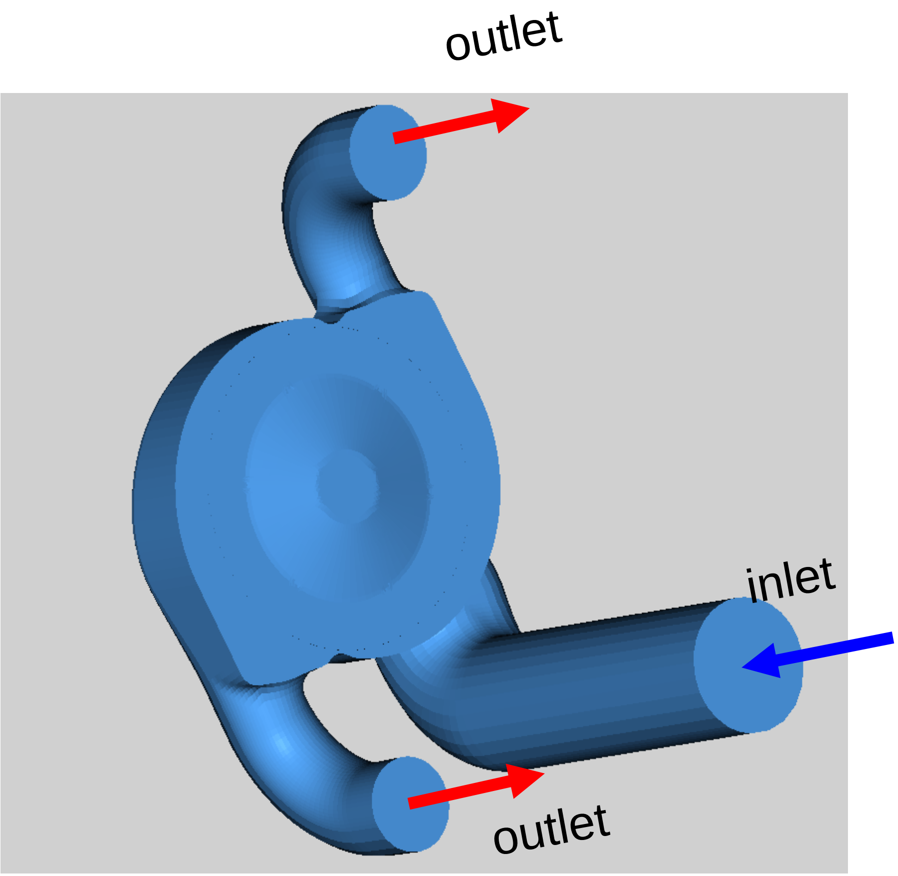
- 100k cells



- Impeller particular



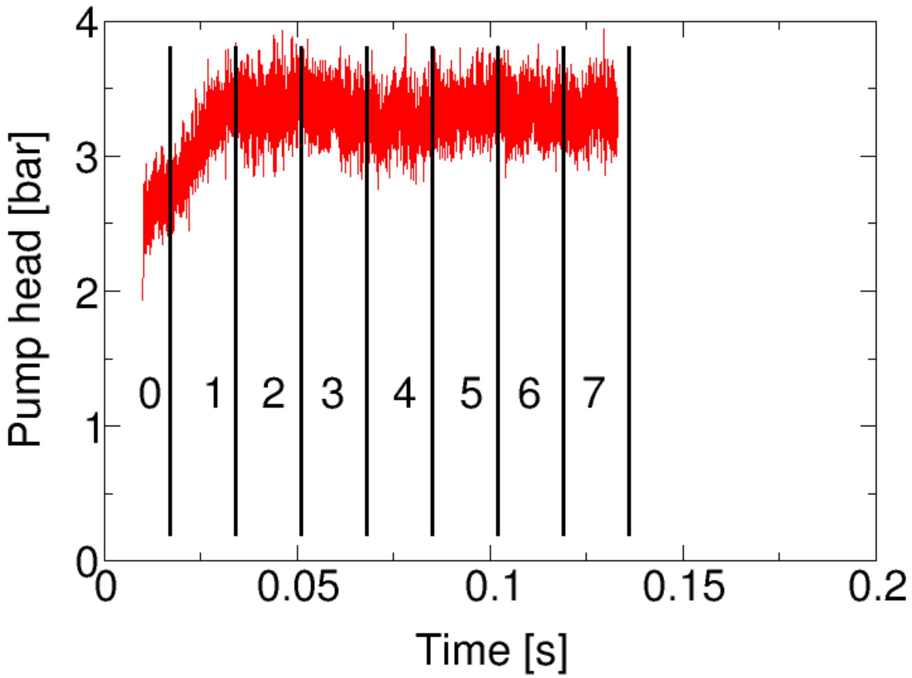
4. Water pump analysis



BONDUARY CONDITIONS

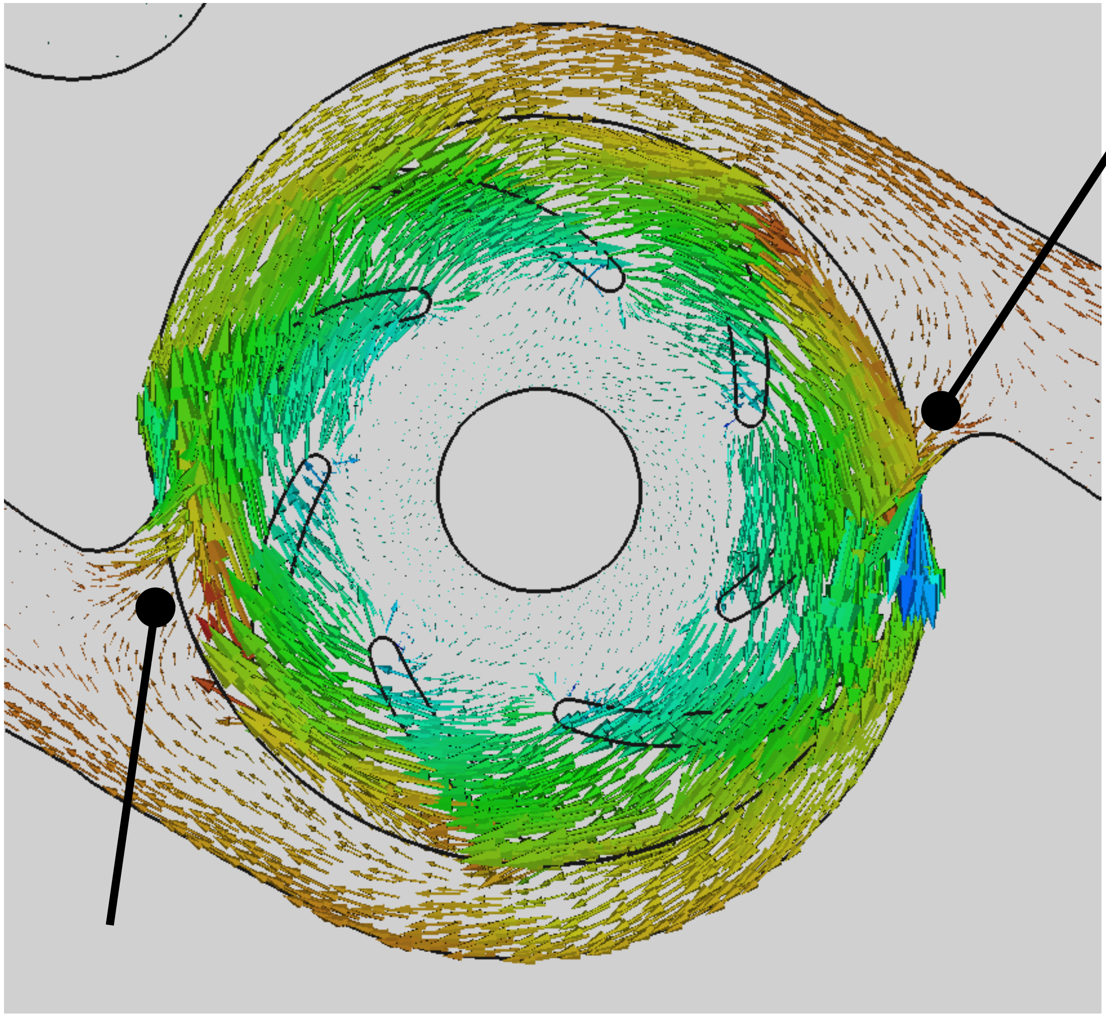
- **Inlet**: flow rate set
- **Outlet**: relative pressure 0 bar
- **Fixed Wall**: zero speed
- **Movable walls**: velocity calculated by rpm rotation of impeller

- Although we've an incompressive fluid, it's necessary simulate some cycles to obtain the convergence of the results.
- This is due to turbolent and to need to achieve the convergence for *k* (turbolent kinetic energy) and ε (dissipation speed of k)



4. Water pump analysis

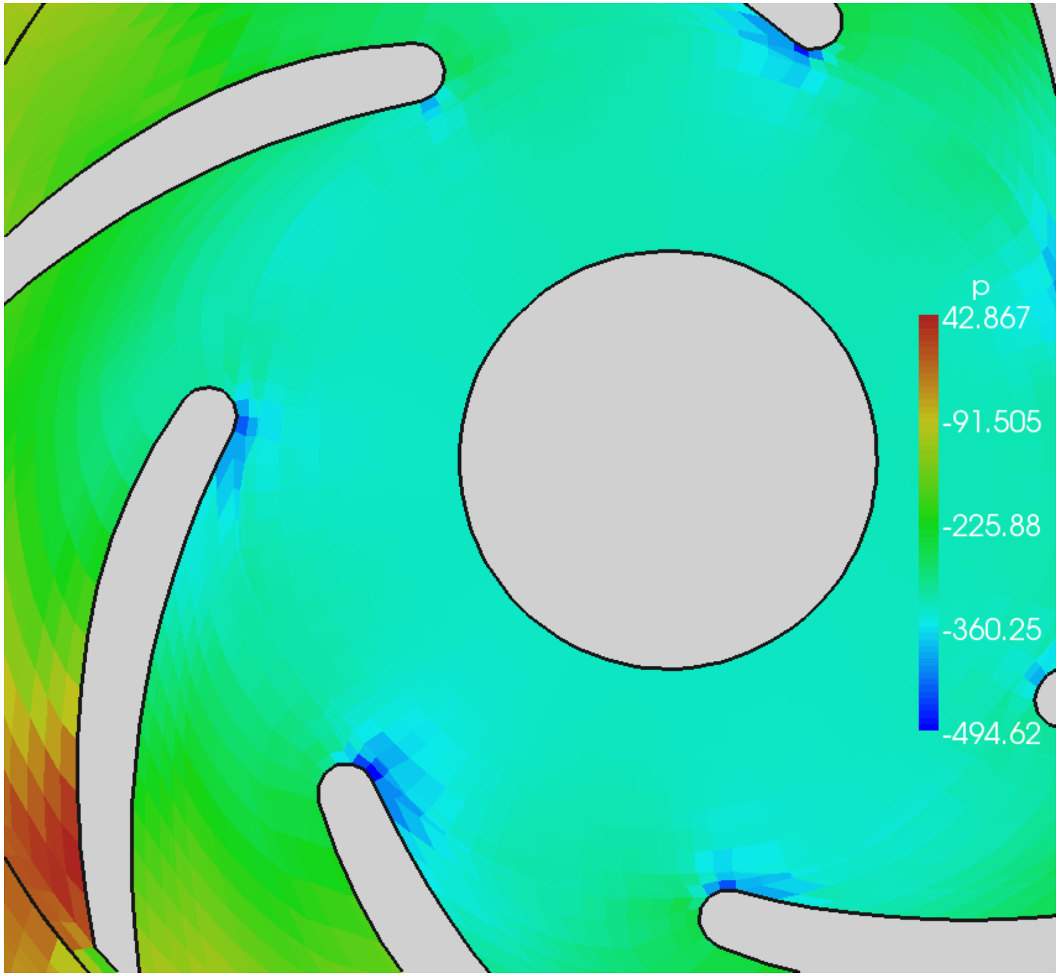
- Flow field on impeller



At the exit of impeller we can see two zone in witch the water flow it’s recirculated. This feature need to be investigated because can limiting the flow rate generated from the pump.

4. Water pump analysis

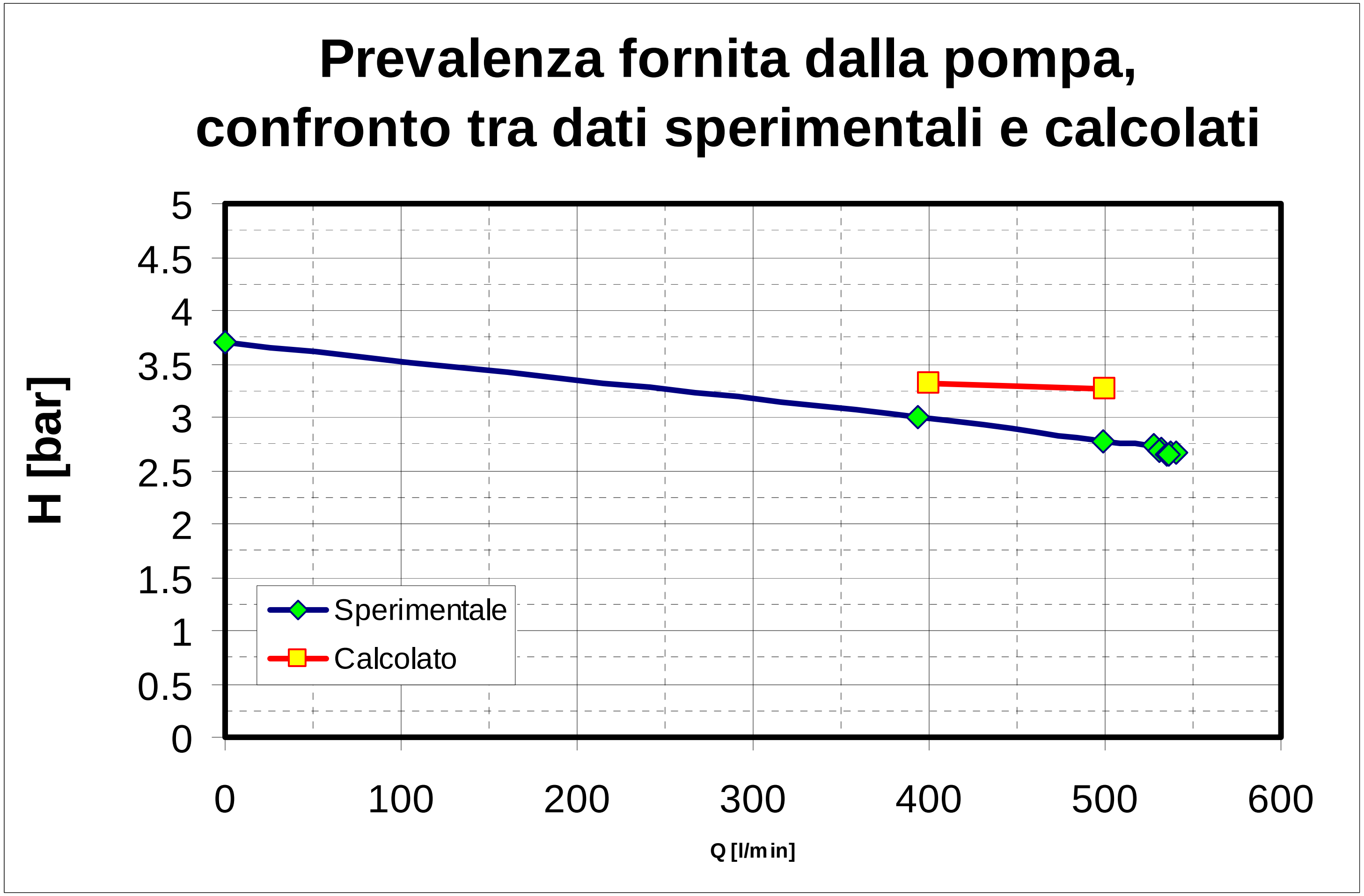
- Pressure field of impeller



Minumun pressure on the inlet area of the impeller

Risk of cavitation

Comparison experimental/simulated



Conclusion

OpenFOAM

- Permit to have an innovative point of view in order to upgrade THE KNOWLEDGEMENT ON ENGINE DURING ITS DESIGN PHASE (increase of tech. dept. capabilities) the consequences of that are:

1. Good Performance
2. Increase Efficiency
3. Time reduction for components development

Thank you for your valuable time

Author: Eng. M. Magni
Seatek Tech. Dept.

11 July 2011