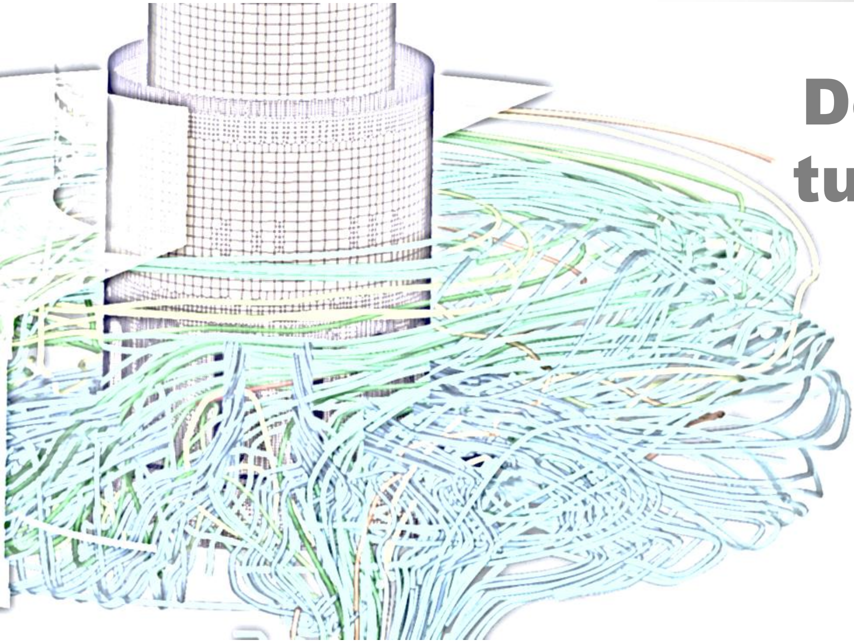
SCS
SuperComputingSolutions
a cineca company



Design improvement of a rotary turbine supply chamber through CFD analysis

SCS Italy ***Roberto Pieri***
Raffaele Ponzini
Andrea Penza

Thesan *Roberto Vadori*
Bernardo Puddu



Abstract

- State of the art
- Market
- Experimental Program@Thesan
    - Prototype and Test Bench
    - Performances and Efficiency
    - Improvements
- Virtual Program@SHAPE
    - Experimental model to CFD
    - CFD results
    - CFD computational performances
- Conclusions and Future works

State of the art

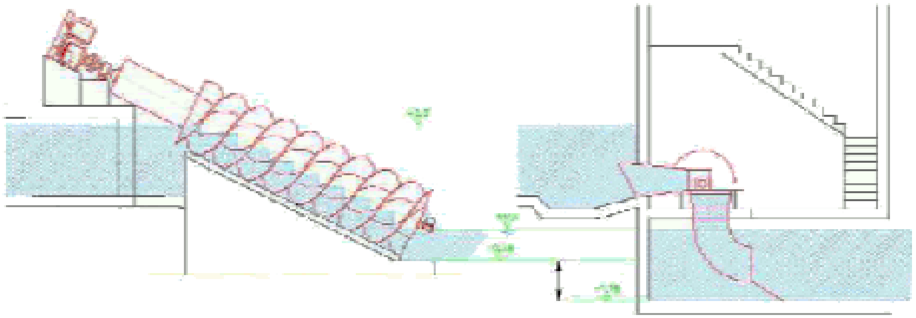
Mini- and Micro-Hydraulic

Mini-hydraulic is a technical term proposed by UNIDO (United Nations Industrial Development Organization) to define hydraulic power plant with an output power less than 10 MW:

As a sub-classification, the following holds:
- **pico plants** P< 5 kW
- **micro plants** P< 100 kW
- **mini plants** P< 1.000 kW

State of the art

Mini- and Micro-Hydraulic



The proposed prototype could be compared with the cochlea, but with some substantial diferences:

Closed chambers instead of free surfaces => **pressure head instead of height only**

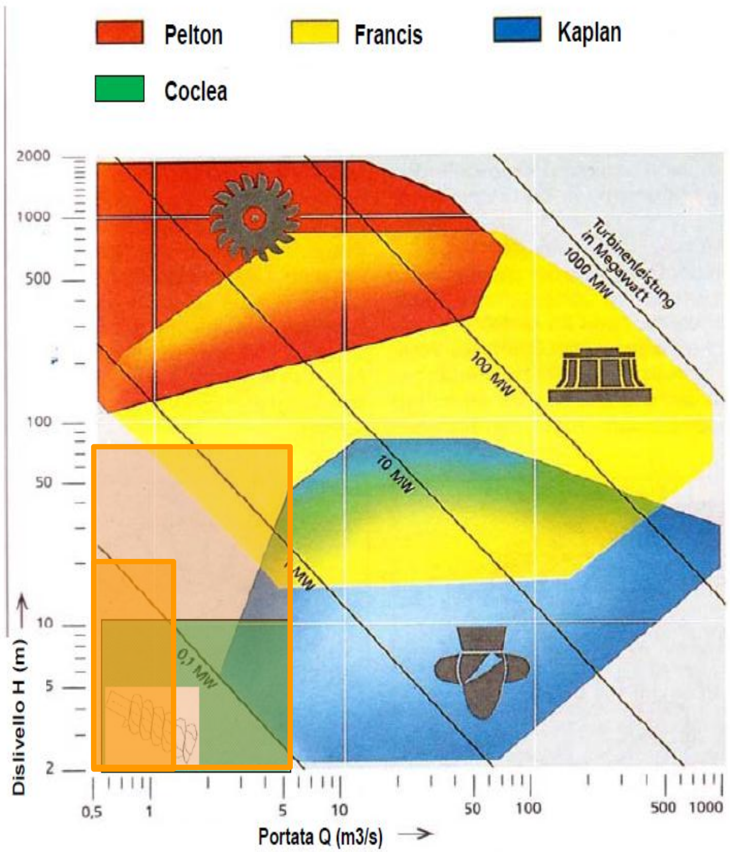
Compact kinematic => **much smaller machine**

Motion is pull out from external rotating chamber=> **built in electrical generator**

State of the art

Mini- and Micro-Hydraulic

If compared with traditonal tubines (Pelton, low flow and high pressure head or Kaplan, high flow and realtively low pressure head) the machine is suited for low pressure head with not-so-high flows. In some sense is a «cheap» machine. Efficiency is then a critical issue.



Design target

Realized prototype

Experimental Program@Thesan

Prototype and Test Bench

- Working physical prototype allows to gather experimental data on integral value

SCS SuperComputingSolutions a cineca company

Experimental Program@Thesan

Performance and efficiency

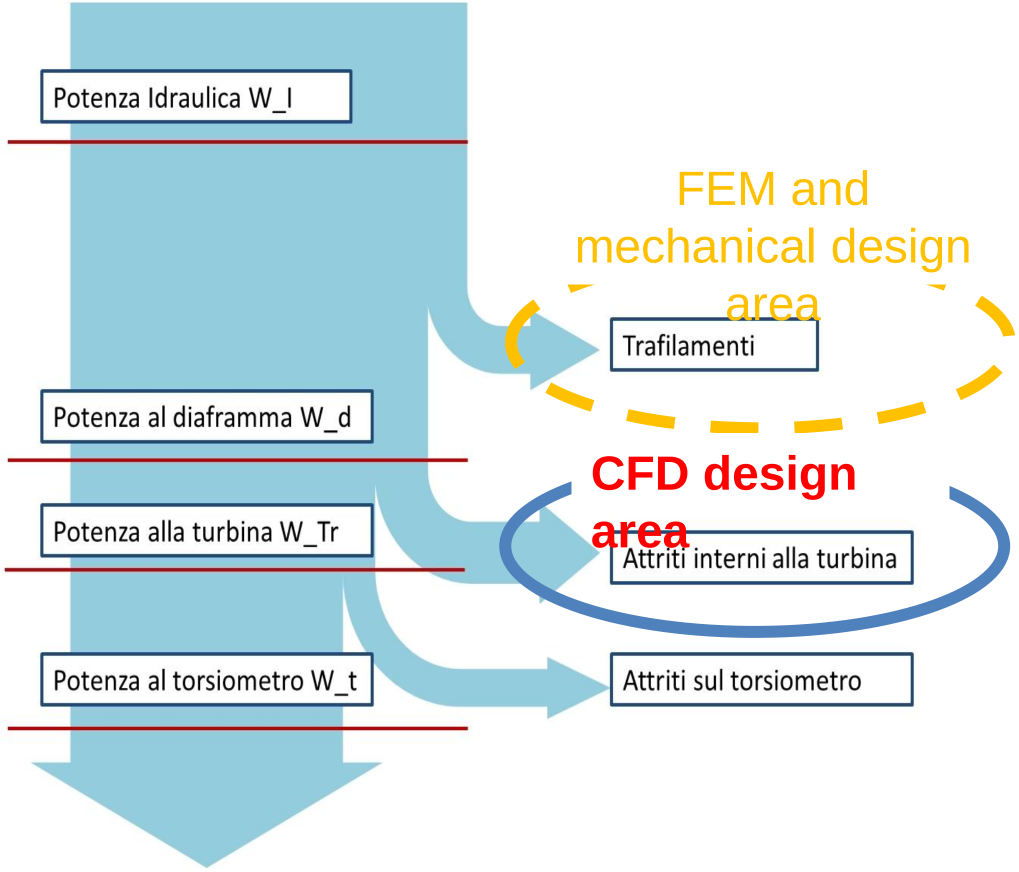
Volumetric efficency: the ratio between the working (hydraulic) power and the supplied hydraulic power

Mechanical efficiency: the ratio between the mechanical power and the working hydarulic power

Global efficiency: the ratio between mechanical power and supplied hydraulic power, the product of the two before mentioned efficiencies.

Experimental Program@Thesan

Performance and efficiency



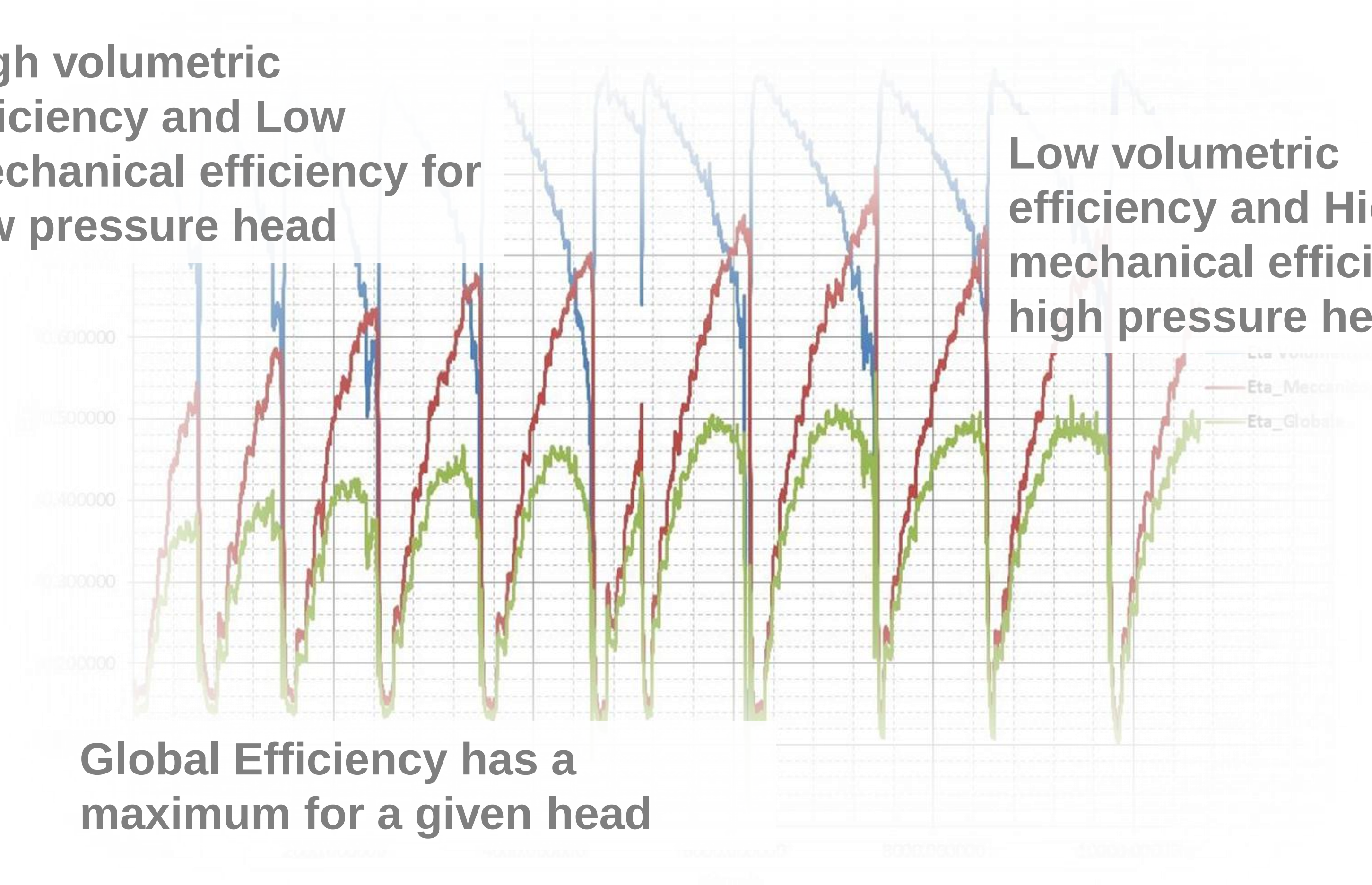
$$\eta_V = \frac{W_d}{W_i}$$

$$\eta_M = \frac{W_{tr}}{W_d}$$

$$\eta_G = \frac{W_{tr}}{W_i} \qquad \eta_G = \eta_V \, \eta_M$$

Experimental Program@Thesan

Performance and efficiency



High volumetric efficiency and Low mechanical efficiency for low pressure head

Low volumetric efficiency and High mechanical efficiency for high pressure head

Global Efficiency has a maximum for a given head

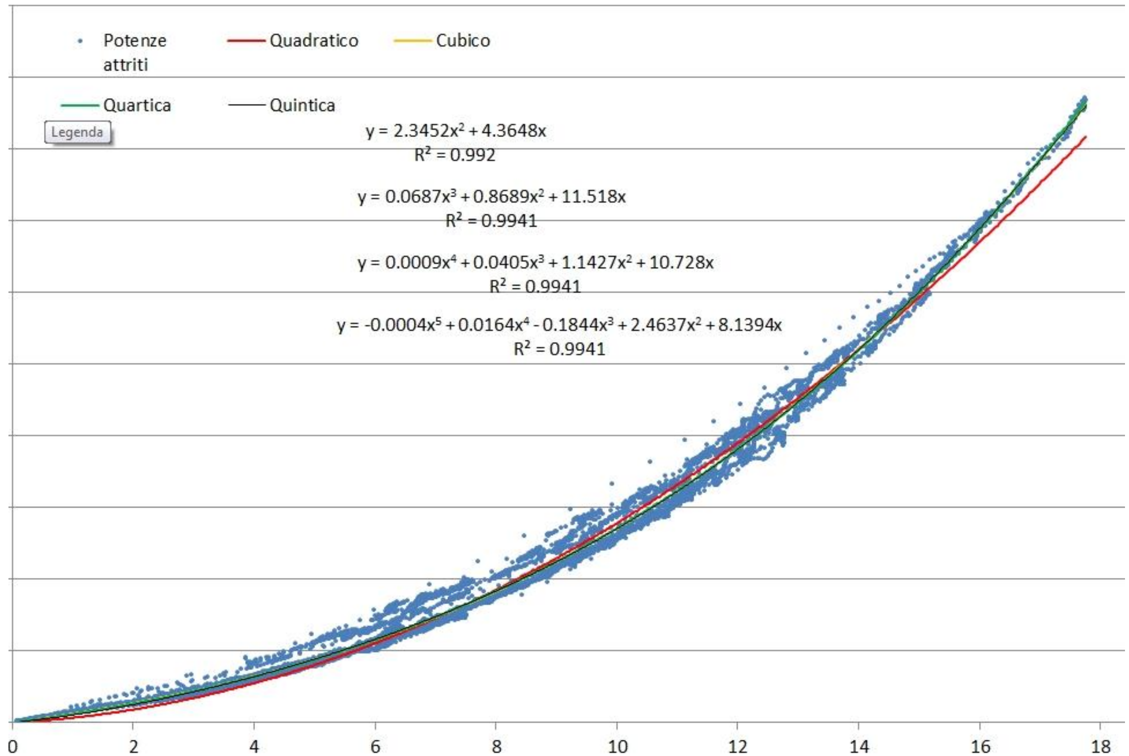
Experimental Program@Thesan

Performance and efficiency

- Loss of mechanical power is proportional to the rotational speed
- The source of dissipating forces is proportional to the square of the velocities.

Questions:

1. Maybe a hydro-dynamic design issue?
2. Could CFD help us to solve the problem?



SHAPE: SME HPC Adoption Programme in Europe

The adoption of HPC technologies in order to perform wide numerical simulation activities, investigate complex phenomena and study new prototypes is crucial to help SMEs to innovate products, processes and services and thus to be more competitive.

SuperComputingSolutions a cineca company

PRACE

SHAPE

SME HPC Adoption Programme in Europe

- SHAPE, the SME HPC Adoption Programme in Europe is a pan-European, PRACE-based programme supporting HPC adoption by SMEs. The Programme aims to raise awareness and equip European SMEs with the expertise necessary to take advantage of the innovation possibilities opened up by High Performance Computing (HPC), thus increasing their competitiveness.
- HPC is a powerful technology that can enable the development of new products or services, reduce time-to-market and cost of R&D or increase quality. The opportunities opened up by HPC are vast and an increasing number of SMEs turn to HPC in order to create new business opportunities.
- The Programme will help European SMEs overcome barriers to using HPC, such as cost of operation, lack of knowledge and lack of resources. It will facilitate the process of defining a workable solution based on HPC and defining an appropriate business model.

Virtual Program@SHAPE

Experimental model to CFD

In order to get a more detailed insight to the fluid dynamics pattern into the prototype turbine we need to:

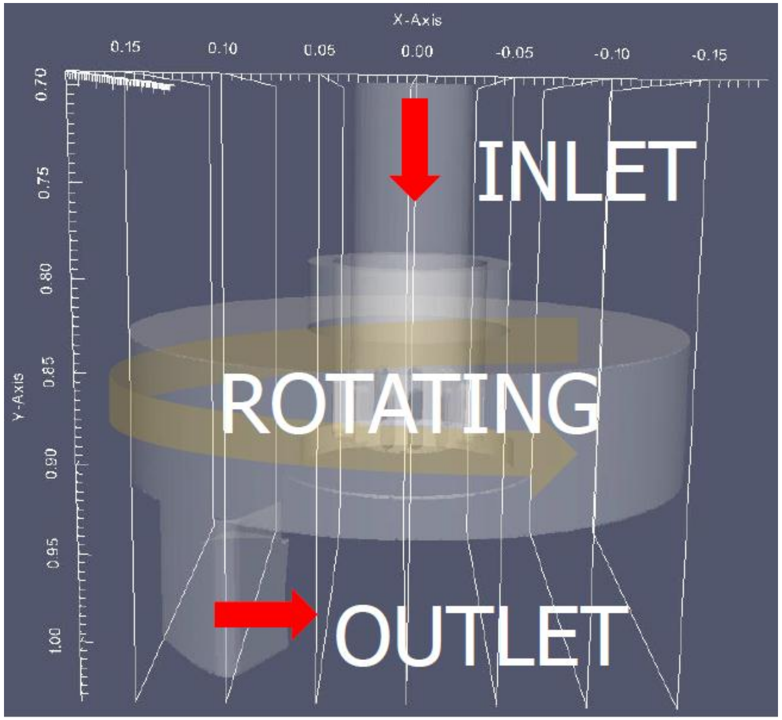
- Build a CFD rotating model starting from the CAD of the prototype device
- Study 4 CFD rotating conditions fixing RPM and Mass Flow Rate at the inlet according to experimental measurements
- Visualize flow patterns to get a better understanding fo the fluid dynamics
- Quantify meaningful fluid-dynamics quantity

Virtual Program@SHAPE

CFD models

CFD modelling characteristics:

- Fluid is Newtonian (water)
- Rotating parts (external tank)
- Turbulent flow behaviour (Reynolds~1e+04)
- RANS modelling (k-omega)
- Moving Reference Frame modelling included
- Software CAD Modelling: Hyperworks (Altair.inc)
- Software Meshing: SnappyHexMesh (OpenCFD Ltd.)
- Software CFD: OpenFOAM 2.3.0 (OpenCFD Ltd.)

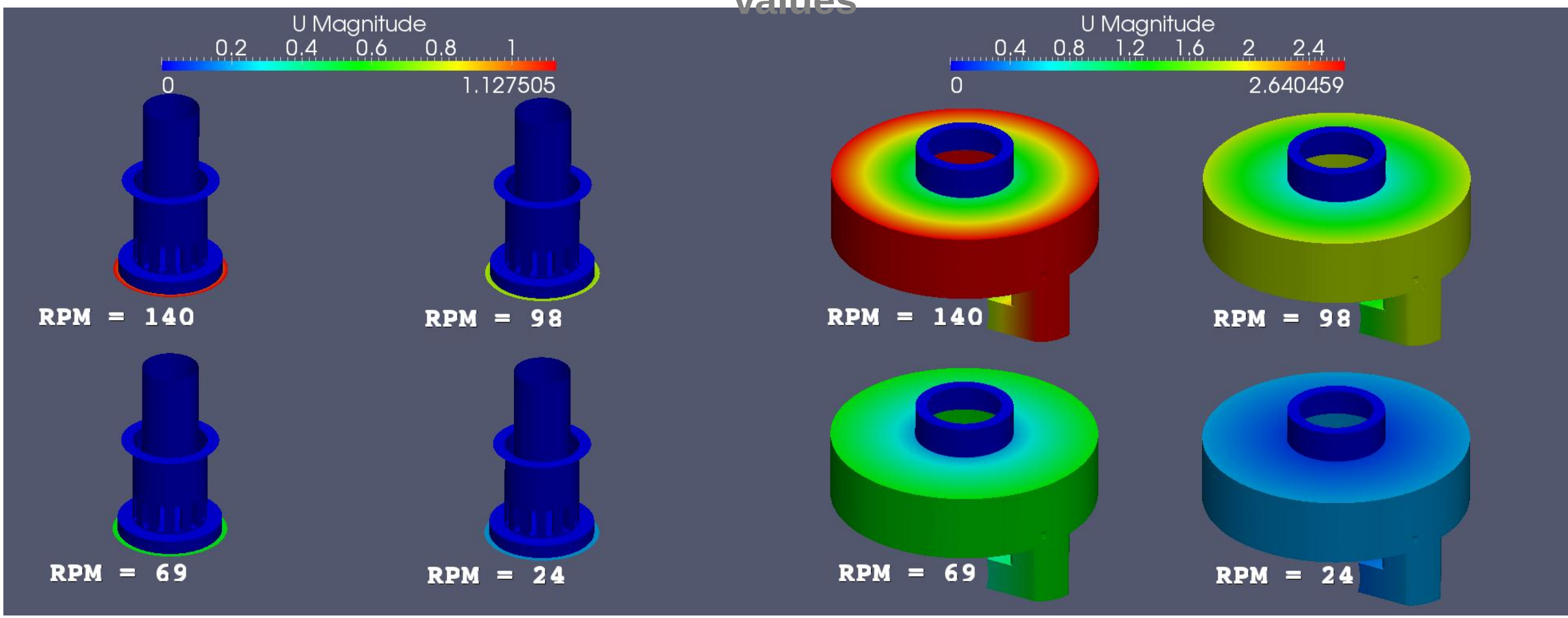


SUPERCOMPUTINGSOLUTIONS
a cineca company

Virtual Program@SHAPE

CFD models

- **Rotating details for BC setting in the 4 models: stator and rotor values**



Stator **Rotor**

Virtual Program@SHAPE

CFD results

CFD model #	Q	RPM	% dP_In_Out
1	1	1	8.03%
2	0,75	0,7	3.96%
3	0,6	0,5	1.81%
4	0,3	0,17	0.36%

(*) All values are normalized

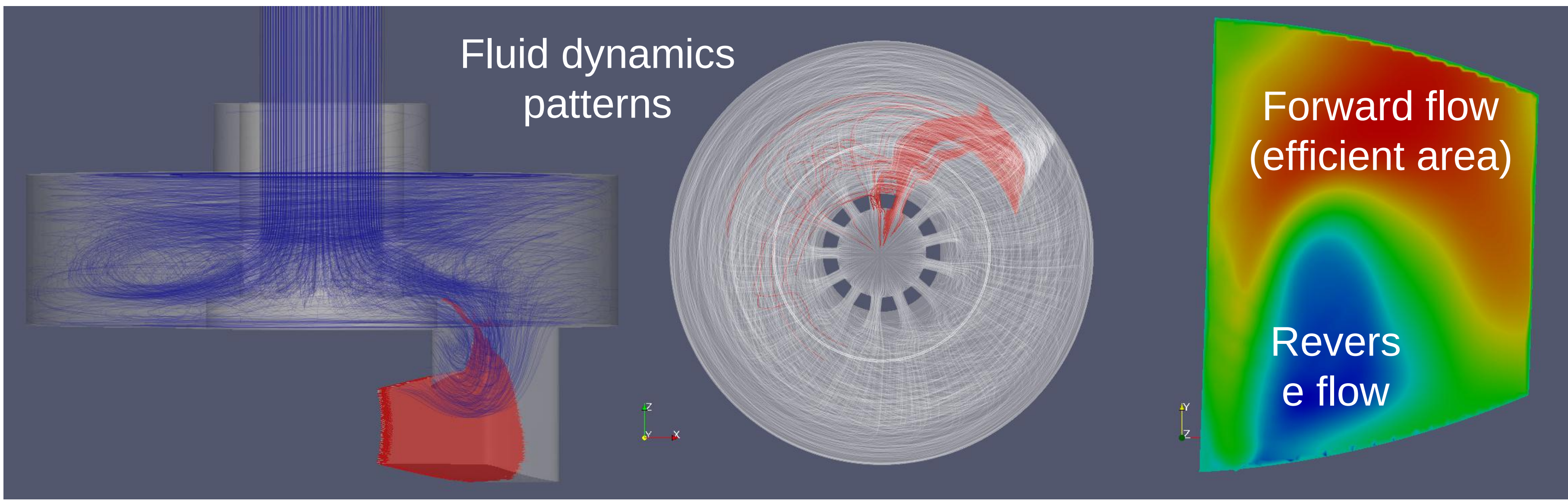
- At higher RPM the device is experiencing an higher pressure loss and higher inefficiencies in total effective outlet area (area with forward flux)
- At lower RPM and flow-rate the device shows a more efficient behaviour. Nonetheless similar trends are detectable.

Virtual Program@SHAPE

CFD results

Flow patterns are coherent to measured integral quantities so that:

- Adverse velocity patterns are present at the outlet section (outlet detail)
- The adverse flow patterns at the outlet are related to a large stable vortex structure at the bending zone of the outlet duct

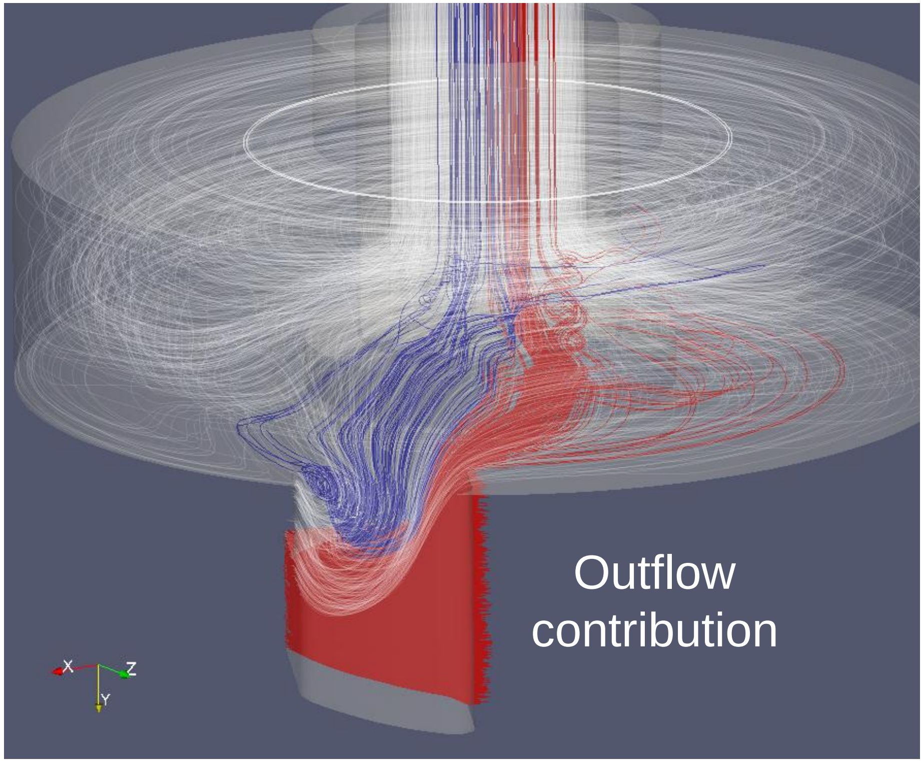


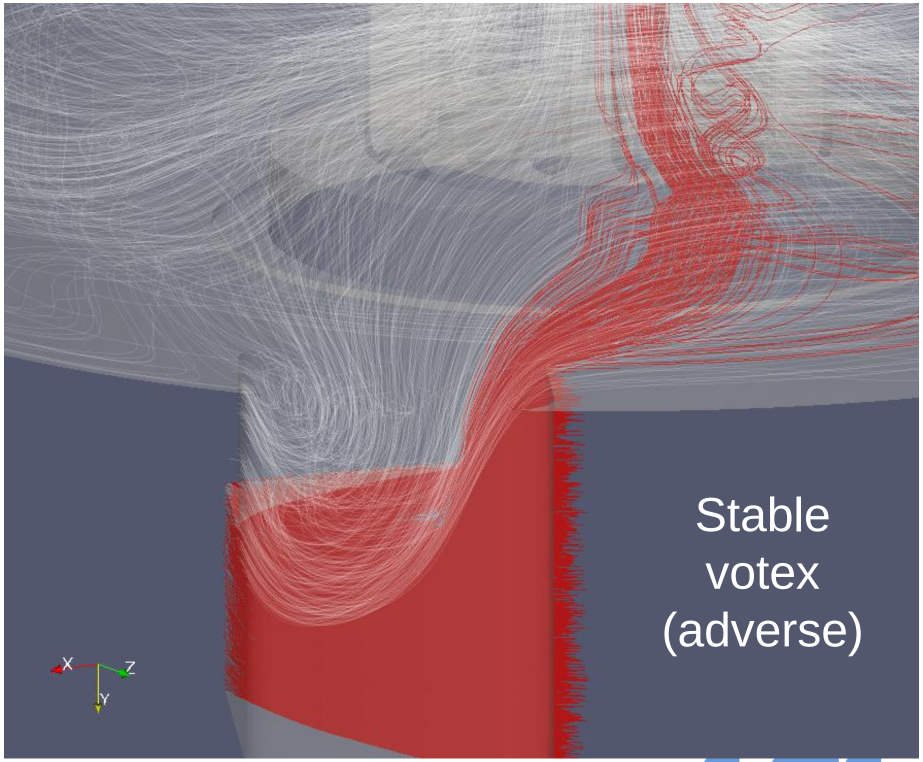
Side view **Top view** **Outlet detail**

Virtual Program@SHAPE

CFD results

Flow patterns are coherent to measured integral quantities so that the most part of the outflow is taken by few slots (in front of the rotating outlet at a given instant) while the greater amount of fluid is experincing a long resident time into the device

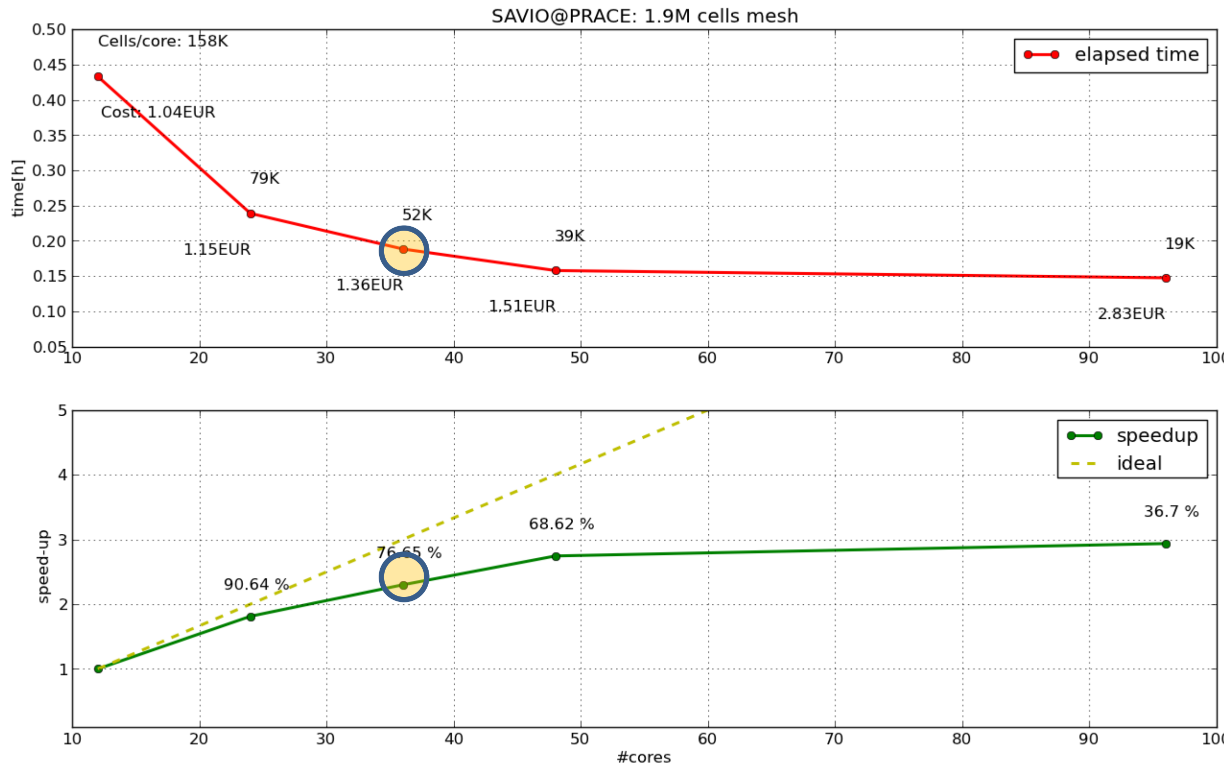




Virtual Program@SHAPE

CFD computational performances

In order to get a more useful evaluation of the gain obtainable with CFD tools when used into an HPC platform we provided standard scalability test for a small mesh configuration (1.9 M cells) and highligthed meaningful quantities such as Averaged unitary computational cost: single loop starting from a stable result, Speed-up and Efficiency



In our benchmark we found:

- an **'optimal' #mesh-cell/core** that can be used for future larger mesh analysis keeping the best scalability (on similar HW)
  - A possible **cost per loop-device** at a given number of cores (0.2 €/core-h) that is very convenient if compared to experimental costs

SUPERCOMPUTINGSOLUTIONS
a cineca company

IDC HPC Innovation Excellence Awards

International Data Corporation (IDC) is the premier global provider of market intelligence, advisory services, and events for the information technology, telecommunications and consumer technology markets.

Conclusions and future work

In conclusion we can state that:

- CFD tools can be very useful in order to get a better understanding of industrial relevant problems when planning to define a new product prototype and when used together with experimental data
- CFD tools are cost/effective with the respect to more traditional experimental tools
- Thanks to CFD we were able to: visualize flow patterns and quantify meaningful fluid-dynamics indices necessary to plan an improved prototype design

In the future the results obtained herein will be used by Thesan to design an improved version of the prototype device.