Virtual Test Bench for Centrifugal Pumps
What is LAPCOS?

LAPCOS™ (Laboratory for collaborative engineering and simulation) is a consortium founded in 2005 qualified in engineering activities by means of advanced computational tools. The company structure is made up of the following companies:
A special partnership: CINECA

LAPCOS has been strongly related to CINECA as partner for HPC (especially for CFD calculations) since its foundation.
LAPCOS services and tools:

<table>
<thead>
<tr>
<th>Service</th>
<th>Tools and Skills</th>
</tr>
</thead>
<tbody>
<tr>
<td>Engineering Mechanical 3D Design</td>
<td>Catia, Unigraphics, Pro-E, Solid Edge, Solid Works, Inventor, Cadenas</td>
</tr>
<tr>
<td>Structural Calculation</td>
<td>Nastran, Ansys, Hyperworks</td>
</tr>
<tr>
<td>Fluid Dynamic and Thermal simulation</td>
<td>Fluent, CFX, Open Foam</td>
</tr>
<tr>
<td>Multibodies Simulation</td>
<td>MSC-Adams , Recurdyn</td>
</tr>
<tr>
<td>Fatigue Behavior Prediction</td>
<td>Fe-Safe</td>
</tr>
</tbody>
</table>
OpenFoam Project:

Why? The use of CFD technology in industry is still limited by the high computational costs, and even more by the high cost of commercial CFD software licenses.

In 2009, LAPCOS in partnership with CINECA relying on the contribute of regional research program starts to explore the Open Foam CFD capability by means of a deep training activity on the code, Linux OS and the programming language.
At the end of 2011 the first project based on Open Foam was delivered

Customer: Pedrollo

Pedrollo is a benchmark brand world-wide for the electric water pump sector. A reputation founded on facilities, technologies, professionalism, and the company’s business model. Pedrollo is a large, constantly growing company that extends over an area suited to a volume of production that has reached **2 million pumps a year**.

The company’s facilities in fact occupy a very large and steadily expanding area that currently amounts to **over 100,000 m²**: dimensions that make Pedrollo an industrial district in its own right, divided into several plants, each with its own efficient structure and position in the production framework.
Problem: improve *time to market* new products

Bottleneck: *High computational time on CFD* (several weeks) due to limited software license and parallel computation

Solution: Customized tool to use *Open Foam* with multi-core parallelization to automate the CFD calculations
**Test bench:** is the most important criteria to check the hydraulic performance for a new pump. In particular to obtain Head (H) - Flow rate (Q) curve and efficiency
Virtual Test Bench for Centrifugal Pumps

**Open Pump**: developed software tool for setting OpenFoam cases and submitting jobs.
OpenPump is a pre-processing and a quantitative post processing software for centrifugal pumps CFD simulations

It works with OpenFOAM-1.6-ext which is its solver software.

OpenPum has been conceived to perform several CFD calculation in a row on the same geometry

**OpenPump aims at providing the analyst with the pumping curve (pressure drop over volume flow) automatically starting from the mesh file and the pumping parameters (rpm and volume flows)**

OpenPump automatically chooses when the solver reaches convergence criteria for the actual calculation by means of a coupling algorithm with OpenFOAM then jumps to the next simulation scheduled.
Virtual Test Bench for Centrifugal Pumps

How OpenPump works

1. Residuals, pressure, torque plots
2. Summary log of the executables launched
3. Summary log of each run:
   - Pressure on inlet and outlet surfaces
   - Torque on rotor walls
4. VTK format of fluid dynamics fields for Graphical post-processing by means of Paraview
Virtual Test Bench for Centrifugal Pumps

OpenPump GUI

- Multicore parameter
- Convergence criteria
- Up to 4 different meshes
- Up to 5 flow per each mesh
- Log windows
OpenPump achievements

OpenPump has been developed to work with a modified OpenFoam executable solver: MRFPisoFoam created by means of modification of the standard MRFSimpleFoam solver that does not allow quick convergence. **MRFPisoFoam provides with a stronger pressure-velocity coupling and has been found more suitable to deal with centrifugal pumps simulations.**

OpenPump is highly user-friendly tool because of its simple Graphical User Interface.

OpenPump allows modifications of its code in order to meet new customer requirements in terms of:

- New OpenPump/OpenFoam-solvers coupling
- Other engineering CFD applications than centrifugal pumps
- OpenPump enormously increases its potential on HPC systems.
Virtual Test Bench for Centrifugal Pumps

Benchmark example (to validate Open Foam results)

<table>
<thead>
<tr>
<th>Accuratezza</th>
<th>ingresso</th>
<th>Uscita</th>
<th>Girante</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFX</td>
<td>Primo ordine</td>
<td>p = 0 Pa relativi</td>
<td>V = 0.43 m/s</td>
</tr>
<tr>
<td>OF</td>
<td>Upwind (primo ordine)</td>
<td>p = 0 Pa relativi</td>
<td>V = 0.43 m/s</td>
</tr>
</tbody>
</table>

ANSYS MESHER
- GGI
- TETRAEDRICA
- 3715612 celle
- Skewness max 0.84
- Skewness media 0.22
Benchmark example

CFX vs OF: Speed field
Virtual Test Bench for Centrifugal Pumps

Benchmark example

CFX vs OF: Pressure field
Virtual Test Bench for Centrifugal Pumps

Benchmark example

<table>
<thead>
<tr>
<th></th>
<th>Torque [Nm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>OF</td>
<td>5.29</td>
</tr>
<tr>
<td>CFX</td>
<td>5.30</td>
</tr>
</tbody>
</table>
Benchmark example (convergence speed)
Scalability Test

Scalability OpenFOAM

- Time [s] @ 30 iterations
- Core n° [x]
Improvements in progress:

1. Implementation of Open Foam 2.0
2. Implementation of Transient Analysis
3. Install Open Pump (jobs manager) on CINECA HPC to perform analysis with 256 core
Thank you for listening

www.lapcos.it
daniele.bucci@lapcos.it

LAPCOS Scrl