EUROPEAN ORGANIZATION FOR NUCLEAR RESEARCH
ORGANISATION EUROPÉENNE POUR LA RECHERCHE NUCLÉAIRE

CERN - TS Department

EDMS Nr:	590571	TS-Note-2004-037
Group reference:	TS-CV-DC/CFD	25 May 2005

A FUTURE FOR COMPUTATIONAL FLUID DYNAMICS AT CERN

Michele BATTISTIN

Abstract

Computational Fluid Dynamics (CFD) is an analysis of fluid flow, heat transfer and associated phenomena in physical systems using computers. CFD has been used at CERN since 1993 by the TS-CV group, to solve thermo-fluid related problems, particularly during the development, design and construction phases of the LHC experiments. Computer models based on CFD techniques can be employed to reduce the effort required for prototype testing, saving not only time and money but offering possibilities of additional investigations and design optimisation.

The development of a more efficient support team at CERN depends on to two important factors: available computing power and experienced engineers. Available computer power IS the limiting resource of CFD. Only the recent increase of computer power had allowed important high tech and industrial applications. Computer Grid is already now (OpenLab at CERN) and will be more so in the future natural environment for CFD science. At CERN, CFD activities have been developed by a dynamic team, consisting mainly of fellows, associates and students. The high turnover of the personnel has unfortunately never allowed the consolidation of a competent and stable team of experts.

A clear definition of these two aspects will lead the future of CFD at CERN.

Presented at the TS Workshop
Archamps, France, May 24 – May 26, 2005

1 INTRODUCTION

Smoke from a car exhaust pipe, a nuclear reactor cooling system, ventilation for cooling electronic systems, dust particle motion in an operating room, fire propagation or wind effect on a sail. All these phenomena, and many others, have one thing in common: they are described by the same physical equations and can be analysed and solved by Computational Fluid Dynamics numerical methods.

CFD allows the development of 3D models and provides numerical solution of thermo-fluid flow problems in confined and unlimited domains. The basis of computational fluid dynamics is the reduction of the continuum differential equations governing the dynamics of the fluid into a system of algebraic equations at a finite number of "grid" points, and the solving of the equations at these limited number of points only.

Since 1993, CFD is used at CERN to support physicists and engineers in all different phases of their projects. In 2004 a team was created in the TS-CV group and charged with all CFD related studies. During this period the Moore's Law on the increase of computer power has been driving the accuracy and efficiency of CFD simulations. The Grid, a new system for distributed computing in development for the LHC project will, definitely also open new dimensions for the CFD applications at CERN as in the industry.

The future of CFD service at CERN is discussed in this paper.

2 COMPUTATIONAL FLUID DYNAMICS

The Navier-Stokes equation complemented with mass and energy conservation conditions describe the flow and the thermal behaviour of fluids. They are a set of non linear differential equations that can be analytically solved only under particular conditions, when they can be simplified (ex. Bernoulli eq.).

Without these simplifications is generally impossible to solve the problem by analytical means. However, progress has been made during the last 30 years by developing a number of mathematical models to facilitate the numerical solution and reduce the calculation time. These models have given rise to a number of sophisticated numerical tools for fluid dynamic calculations. Some of these tools have developed further into commercial packages of which Fluent, StarCD (the one CERN selected) and CFX are the most important products currently on the market. These codes are now widely used in many industrial, high tech and research applications.

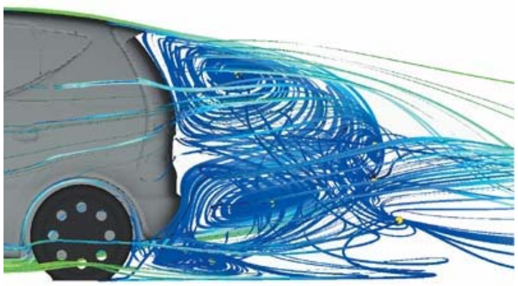
3 CFD APPLICATIONS

A description of common industrial applications is not within the scope of this document but a short overview helps to understand the wide application field of this technology. The second part of this chapter focuses on CERN applications.

3.1 Industrial applications

We will list here the major industrial and high-tech applications of CFD classed by industry. The informed reader can skip the entire §3.1 chapter and go ahead to the CERN applications.

The Automotive Industry is perhaps the most important user of CFD techniques because its needs are complex and varied. The need for rapid prototyping and use of analytical tools during all stages of design and development, plus the tight schedules to reduce the time-to-market create one of the most challenging environments for CFD. External aerodynamics, under-hood air flow and thermal management, induction and engine in-cylinder flow, fuel injection and combustion, climate control and passenger comfort, aero-acoustic noise prediction are the major fields of application. Famous examples can be found in the Formula 1 sport domain where all major racing teams have one of the CFD code developer company as a strategic supplier.



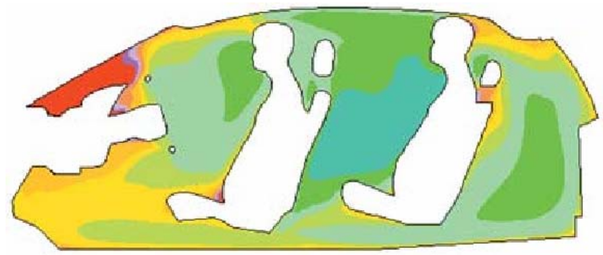


Figure 1 - Car aerodynamics **Figure 2 - Car internal comfort simulation**

A second important field is civil engineering. CFD helps engineers, architects and project managers to acquire a better and more detailed understanding of the issues involved in heating, ventilation and air conditioning (HVAC). They can also be used in the design of clean rooms and buildings that protect personnel from biochemical and radioactive agents, and for smoke and pollutant movement and fire spread prediction within buildings and tunnels. Wind loads on civil structures (bridges, buildings, etc.) are easily calculated with CFD techniques. Figure 3 shows the effect of light regular wind on the Takome Bridge: Von Karman vortexes created a resonance effect that was responsible for the collapsing of the bridge in 1940.

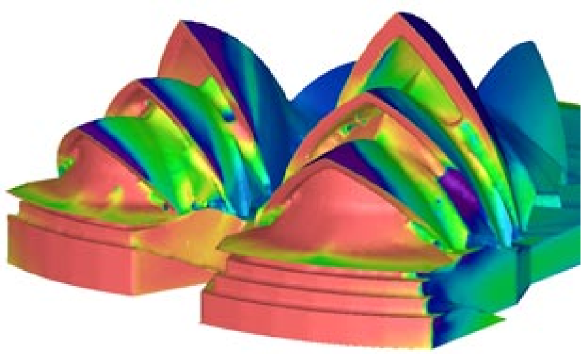
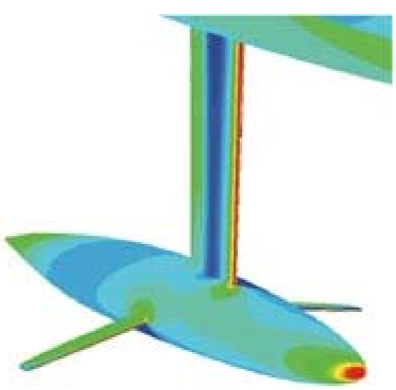


Figure 3 - Sydney Opera Hause CFD model **Figure 4 - Takoma bridge failure (1940)**

The marine industry's flow simulation needs are diverse and challenging, spanning many scientific disciplines. CFD is used in a range of applications such as: wind and wave loading on offshore and underwater structures, oil and pollutant dispersion, propulsion system optimization and more. Another high technology example is the Alinghi project that extensively used the resources of the CFD department of EPF of Lausanne firstly to design the sail-boat and secondly to refine, on a daily basis, its performances during last America's Cup.



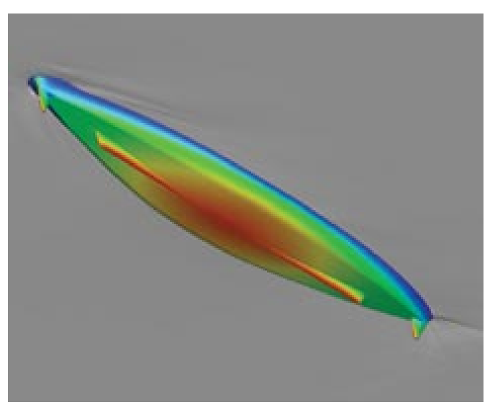


Figure 5 & Figure 6 - Pressure map of American's Cup sail-boat keel

Another good example is found in the medical sector. CFD is especially used in the cardiac area to perform "hemodynamics" simulations. The fluid mechanics of the blood flow has been found by medical and biological researchers to be the key factor in the pathogenesis of arterial disease.

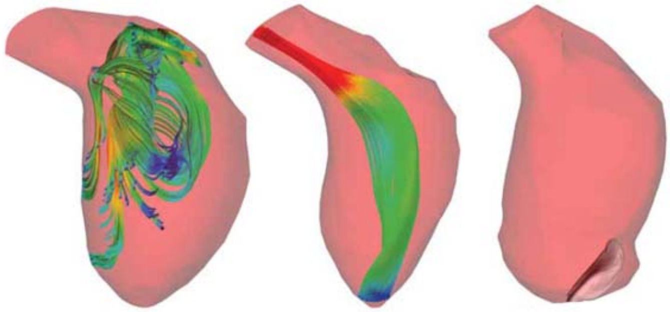


Figure 7 - Blood streamlines at heart filling and ejection cycles

Handling of complex flows and chemical reactions, mixing of fluids and processing of waste products are some of the areas where CFD analysis can help the chemical and process engineers. An optimization of a blast furnace temperature distribution can e.g. improve the quality of the steel produced.

CFD thermo-fluid models provide as well great benefits in the turbo-machinery industry where detailed experimental data can be very difficult and expensive to obtain.

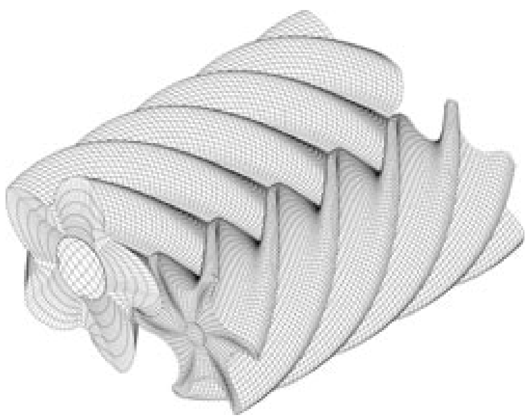


Figure 8 – Screw compressor 3D model

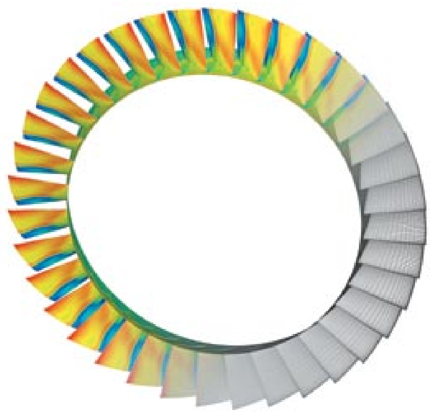


Figure 9 – Gas turbine stage model

Finally, the electronics industry is more and more faced with cooling problems and CFD simulations can speed up the development phase, reducing the number of real-life tests to be performed. This is more and more important in a sector where time-to-market has become a fundamental parameter of performance for company and its products.

3.2 CERN applications

Large part of CFD applications at CERN are concerned with thermal convection problems: natural, forced or mixed convection effects in and around detectors or sub-detectors. A non exhaustive list is presented below in a "client" order list.

3.2.1 CMS

Since 1994, already during the proposal studies, CMS used CFD simulations to study the thermal behaviour of the silicon tracker [1]. A 2D study was carried out to determine the temperature distribution inside the silicon tracker under natural convection conditions and different nitrogen forced ventilation configurations.

More recently another 2D study was performed to understand the evolution of humidity concentration inside the inner tracker TOB detector. In the transient model, the volume, initially filled with air at cavern conditions, is flushed with the dry nitrogen flux. The simulations provided important information on the best position for the nitrogen inlets and outlets. They gave also an approximate indication of the time needed for flushing before decreasing the tracker temperature to avoid dangerous condensation [2].

3.2.2 Alice

The technical coordination of the Alice detector is one of the most important clients of the CFD service team. The value of the CFD simulations is clear for them and the simulation results have been often the base for important technical decisions.

The entire ventilation strategy of the L3 magnet has been tested with a 3D numerical simulation to define the best temperature distribution. The results of this study (see Figure 10) helped the Alice community to decide implementing of a thermal screen to protect the TPC detector that needs extremely stable thermal conditions [3].

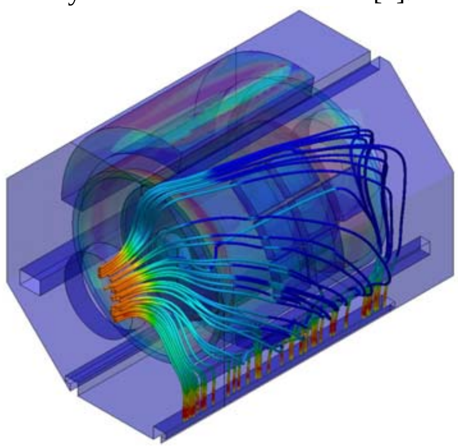


Figure 10 - Alice L3 magnet ventilation

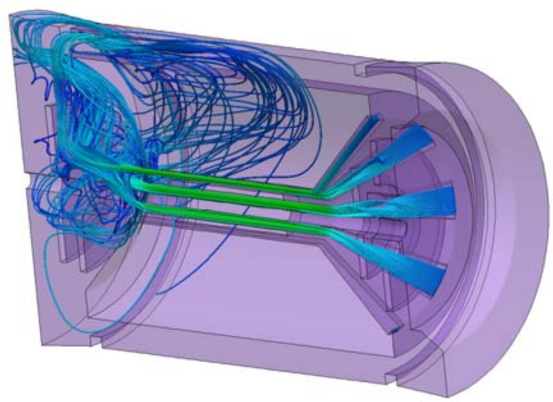


Figure 11 - Alice ITS cooling

The Alice inner tracker temperature map (ITS) has also been investigated many times (see Figure 11). In Alice the heat power produced by every sub-detector shall be removed by the sub-detectors own cooling system. Cooling methods and performances shall be sharply verified. Temperature maps under forced air ventilation conditions aided in the location the hottest point and to optimize the configuration of the ventilation system [4].

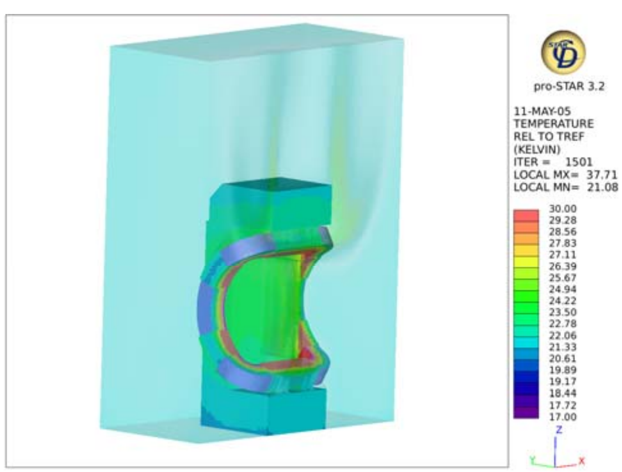


Figure 12 - Temperature map of Alice Muon Simulation

Figure 13 - Alice Muon Detector

Alice Muon Detector natural convection heat transfer has been recently studied with a 3D CFD simulation to define, and in case modify, the amount of heat released to the cavern (see Figures 12 and 13). The result of this study supported the decision to not install of a supplementary thermal insulation on the coil [5] during the final installation in the cavern.

3.2.3 Atlas

The air humidity problem has also been investigated for the Atlas inner tracker, where temperatures as low as -25 ºC are foreseen so that condensation (icing) can have dramatic consequences on the detector life. In this case the detector will be flushed with CO_2. CFD results showed that after about 2 hours of CO_2 flushing, the Inner Detector will reach the sufficiently low dew point to allow the cooling system to start lowering the distribution temperature [6]

Another important 2D transient simulation has been recently done on the Atlas Muon chambers system in collaboration with the Russian institute of RFNC-VNIITF (Snezhinsk – Russia) [7]. CFD studies evaluated that natural convection will not be sufficient to remove all the heat dissipated by the electronics on board of these detectors. Very high temperature spots (not compatible with electronics) have been detected. As a result dedicated active thermal screens have to be installed in strategic positions (namely where the air circulation meets obstacles) [8].

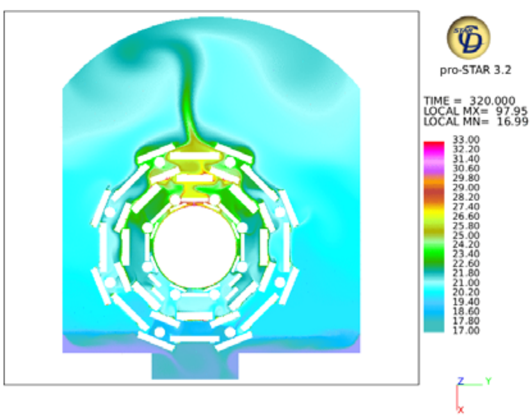


Figure 14 - Temp. map of Atlas Muon Simulation

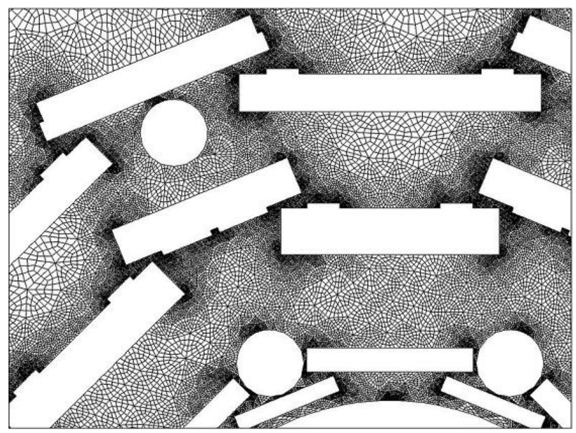


Figure 15 - Detail of the ATLAS Muon mesh

3.2.4 LHCb

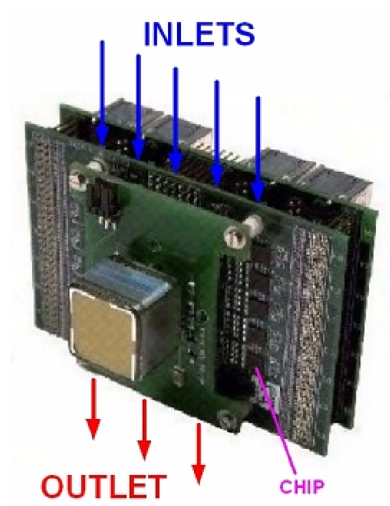


Figure 16 – LHCb PS electronic card

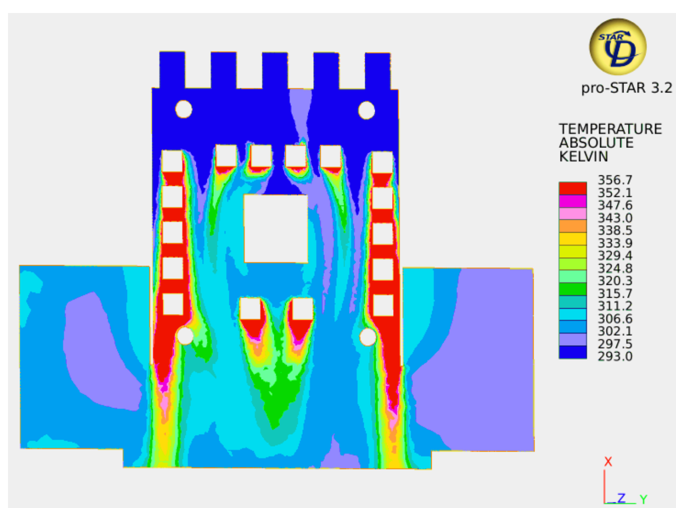


Figure 17 – Temperature map under forced ventilation

LHCb experiment offers an important example in the field of micro electronic cooling. A 3D model of a forced air cooling system of PS and SPD electronics board is currently being developed (see Figure 16). In this case an appropriate air cooling system must be designed to limit the temperature of the electronic chip which are not allowed to exceed a specific value under the given conduction and convection conditions (see Figure 17). Results are not available yet.

3.2.5 CNGS

The air cooling system of the horn and reflector devices is of critical importance to their performance. The supporting aluminium structure is required to remain under 100ºC to avoid variation of the mechanical characteristics of the metal which may otherwise compromise the geometry of the system and the tight alignment of the equipment with the beam axis. Shielding walls, heated by radiation energy deposition, create a sort of closed "oven" where external cooling air flow cannot easily enter (see Figure 18). In this case the CFD simulations (see Figure 19) are of great importance as the concerned area will not be accessible once the first run takes place due to the high radiation levels. All "final" configurations must be defined now. Since prototyping would be extremely expensive to carry out, a number of CFD simulations are made now, during the final construction phase, providing important information on the shielding structures for the optimization of the thermal exchange [9].

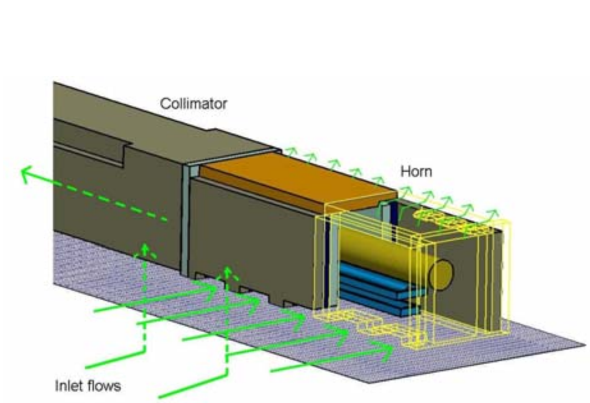


Figure 18 - 3D model of the horn and its shielding

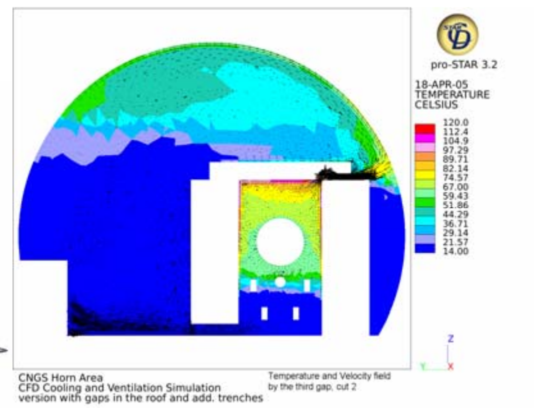


Figure 19 – Temp. map in a target chamber section

Another example is a safety study performed using the 3D model of the CNGS structure. This project studies the consequences of the rupture of the titanium windows that close the 1 km long and 2.5 m of diameter low pressure (100 Pa) pipe downstream of the target chamber. It was shown that the large empty volume would be filled by air coming form the surface.

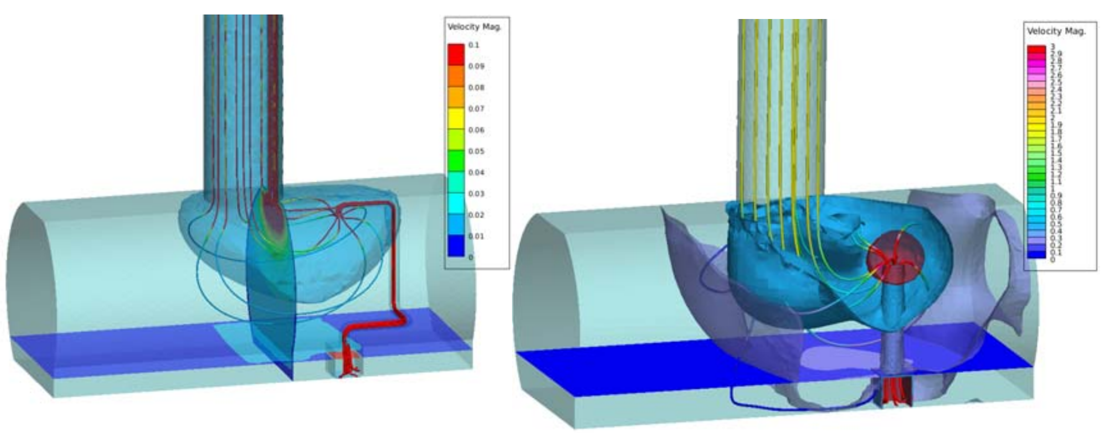


Figure 20 - ECA4 cavern: air streamlines and velocity field with or without the safety duct

A supersonic air flow would pass through the CNGS underground buildings. In particular this flow would affect the ECA4 cavern that is normally accessible. With the help of the CFD simulation a dedicated air duct has been designed to move the high velocity point to the ceiling region of the cavern (see Figure 20) [10].

3.2.6 Others

The complete list of examples is so long that we limit our description to some other interesting cases:

- 3D simulation of the air conditioning distribution on the new LHC computer hall of building 513 (2003).
- The Fast Energy Amplifier project of Carlo Rubbia used 3D CFD simulation to understand the natural convection cooling of the beam target in a circuit of liquid lead (1995).
- The fire evolution inside the Innovation Globe (under development).
- The thermal behavior of the SPS warm magnet under forced water cooling and external air convection (under development).

4 NUMERICAL MODELS ADVANTAGES

CFD is a non-intrusive research method that can provide insight into fluid flow problems that would be too expensive or physically impossible to explore by experimental techniques alone. In the latter case the numerical investigations help to define the best configuration of the physical model that eventually will be tested in real conditions reducing the prototypes number and cost. The insight and understanding gained with CFD simulations give added confidence to design proposals with a reduced risk, avoid the need to design by over-sizing and over-specifications, reducing drastically the time and consequently the cost of the prototyping phase.

At CERN, most of the time, the final system is the prototype. CFD simulations are often the only way to estimate the behaviour of a system before it will be constructed. CFD allows many investigations that would not be possible to realize on a physical model because of lack of time and resources.

Improvements in computer hardware performance, at an ever lower cost, combined with the introduction of parallel processing and load sharing technologies, have dramatically reduced global time for model simulation. These advantages will become even more manifest during the next years due to the increase in calculation power and the advent of GRID systems that will allow achieving faster solutions of larger, more complex model problems.

5 CFD TEAM

5.1 Present situation

The CFD activity started in the TS/CV group around 1993 when a technical student was appointed to work 100% on these studies. Since then a number of young engineers (technical students, fellows, PJAS, UPAS, students coming within Spanish and Portuguese programs) have participated in CFD activity spending short to medium periods at CERN.

In 2004 when the TS department was created, the CV group decided to structure this activity into a formal service team. With 6 young engineers working on CFD problems a team was created in the Detector Cooling section. With the contribution of all members of the newly formed team many problems have been analyzed and some solutions implemented.

5.1.1 The team

A team building effort was necessary: all the persons were used to work independently, sometimes in separate and distant offices with knowledge sharing on a voluntary and friendly base. The entire team has been move to the same building. Regular weekly meetings started, with open discussions on the advancement status of every project and on technical problems.

Some basic quality procedures were introduced to manage the service's projects from the "client" request to the final report [11]. This simple procedure had an important impact on the team activity: firstly it forces the "client" to clearly define the problem and to provide all the relevant information needed for the study. Secondly it defined the "closure" (formal termination) of the study with a final standard technical report. The description of the problem, the model construction, the results obtained, conclusions and recommendations for further studies are now clearly defined and stored for the future.

5.1.2 Archiving

All information regarding previous studies were previously stored in different formats and spread in local disks, different document servers, or personnel web pages. The team members invested some time to collect all past document relevant links and have stored them along with all new documents produced in a dedicated EDMS structure ("TS-CV CFD Studies"). Knowledge sharing has been recognized as key factor for the team life. A CFD Web page (cern.ch/CFD-studies) has been created to facilitate the information and knowledge transfer and to allow a structured and direct access to the final results of past CFD simulation studies.

5.1.3 Training

There are two different areas where training is fundamental for the team: the induction training of new members and the continuous technical training.

The induction training is important to reduce and optimize as much as possible the time needed for a new member to be independent and start giving positive results. The member turnover is, in fact, extremely high. Each new member takes between 2 to 4 months to start using the CFD tools properly, independently and efficiently. He starts being productive in the following period but, in some cases, he leaves CERN at the end of his contract which lasts an average of 12 months.

In this case as in others, the CFD web site is an important tool to share basic information and technical tips not only with present colleagues but also with past and future users. A dedicated "user area" has been created to list and collect basic questions, tips to solve computational problems, training documents, external links to existing CFD forums and listing of relevant books and articles on the CFD subject.

The second area is the permanent technical training. Computational Fluid Dynamics is a wide subject of study that requires considerable expertise. More than pure fluid mechanics competences, proficient CFD use also requires numerical analysis knowledge too. The CFD engineer needs a deep understanding of the different mathematical models available to the different flow situations. Only after a few years of experience on CFD problems, the engineer really starts to understand how much more he needs to learn on this field to become a true expert. Experience and continuous training are the keywords for a professional CFD engineer.

To cope with training needs a partnership has been created with the "Centro Interdipartimentale di Fluidodinamica e Idraulica" of Udine University (Italy) and a 3 day training session on theoretical and practical CFD aspects has been organized [12].

For the technical training on the Star-CD package and the technicalities related to its new tools CD-Adapco provided support to the team with a dedicated training especially related to the new 3D modeller facility (Star-Design).

5.1.4 "Advertising"

The future size of the CFD team is related to a clear definition of the potential workload that can be expected from CERN and Experiments. For the time being project requests and related work come and in part still comes to the service as a result of personnel contacts. Those who have understood the added value given by the CFD results keep using its services, sometimes intensively.

To obtain a more stable workload and justify any future investment, the team must widen the basis of potential "clients". Questions that arise in this context are:

- How many activities and which at CERN could profit from the CFD service?

- How many persons are not aware of the possibilities offered by CFD?

The answer is not straightforward but a number of initiatives have been organized by the team to promote its activity. Two seminars have been held during last year [13] [14] and others are under preparation. Some leaflets have also been posted in the main CERN notice boards [15]. And the first page of the Bulletin will be soon dedicated to the CFD team activity.

5.1.5 Computer power

As discussed in the previous chapter available computer power and job turn around times are limiting factors of CFD studies. The CFD numerical simulations can “saturate” all available computing power that is made available. The major processor chip manufacturers regularly ask the CFD tool suppliers to test the performance of their new hardware device. Some other questions remain:

- How many times CFD has not been requested because it takes too much time to give results?
- How many times CFD has not been requested because the accuracy of the results was insufficient?
- What could be the benefit of a “one week answer” during the LHC experiments construction phase?

New computer resources for calculation have been required from the IT department by TS, with the support of the Technical and Computing Coordinators of the four LHC Experiments [16]. This request triggered a series of performance tests of the CFD code on the OpenLab Itanium cluster. These test proved the efficiency of the Itanuim+Infiniband cluster; it is nearly 4 times faster than the standard LxParc cluster where 6 dual CPU machines are available for the team. The same tests showed a “scalability” factor that remains acceptable up to 32 calculation nodes.

After these tests have been done the IT department is now offering a dedicated cluster built with 20 OpenLab double CPU machines. These 40 new nodes will be added to the 12 existing ones on the LxParc cluster. The new cluster will be available soon and will allow a large improvement of the solution speed of the running and future projects. This is already a great opportunity to demonstrate the value of the CFD service.

5.2 Opportunities

The future of the team depends on the future evolution of CERN projects and on the improved efficiency of the simulation studies. This last factor can be boosted by acting on two different levels: computing resources and human resources.

5.2.1 LHC Grid

CERN will soon define the large scale configuration for the LHC Grid project. This new computer power and data storage capacity sharing service has the aim to turn the global network of computers into one vast computational resource. It will have its breakthrough soon and the impact on a number of different fields could be comparable to the development of the Web in the ’90.

The Grid will become soon THE natural computing environment for CFD applications. CERN CFD team could have an opportunity to run simulations in this environment with an important advance and advantage over the industry!

This development will be possible only in partnership with CD-Adapco, Star-CD supplier. They should be involved in the development and testing of its product in the next generation GRID computation environment. This is a completely new direction that still has to be developed.

5.2.2 Experience consolidation

As discussed in §5.1.3 the continuous rotation of young engineers is limiting the average competence level of the team. CFD is a complex science and uses complex solution techniques. A deep professional competence on this field can be achieved only with a long term training strategy which unfortunately is not compatible with the present nature of the contract of the team members.

Additionally, the TS department has already used its Fellows and Students quotas in advance of the next years, thus these resources will not be available to CFD team anymore. The Spanish, Portuguese programs have an average turnover even higher than those of the Fellow and Students.

In order to maintain at least the CFD framework a dedicated staff position with the mandate of technical coordination of the team is required. This would permit the concentration and later distribution of the CFD training efforts that are required to maintain and increase the service level.

6 CONCLUSIONS

The description of CFD science and its application in industry and at CERN has been presented. The added value of CFD simulations in many CERN applications, especially in the Experimental area, has been clearly shown.

The evolution of the CFD activity at CERN during the last 13 years has been presented with emphasis on the action taken during the last years to develop the service to its present status. Further opportunities for an important development of the CFD service at CERN have also been presented.

The future of Computational Fluid Dynamics at CERN is now depending not only on the past, present and future performance of the team but also on the support of the TS and IT departments to invest in the necessary human and computing resources.

7 ACKNOWLEDGEMENTS

I wish to thank all members of the present CFD team: Sara Correia, Moritz Kuhn, Anna Mueller, Antonio Romanazzi, Vaclav Vins, Izabella Wichrowska-Polok. They performed a number of simulations described in this paper and provided all the CERN related impressive images.

REFERENCES

[1] CMS - The Compact Muon Solenoid - Technical Proposal CERN/LHCC 94-38 - Page 40.

[2] Nicola Gatti - “Moisture Removal in the Cms Tracker Outer Barrel Using Dry Nitrogen Flow” - EDMS 503499

[3] Anna Mueller - “The CFD study of the Alice L3 volume ventilation” - EDMS 528133

[4] Anna Mueller - “Alice ITS CFD thermal simulation” – EDMS 592558

[5] Antonio Romanazzi - “Temperature Field on Alice Muon Detector” - EDMS 528923

[6] Izabella Wichrowska-Polok - “CO2 circulation inside ATLAS Inner tracker detector” - EDMS 591951

[7] Arsentiev A.P., Kiselev I.V., Lapaksin A.A., Lebedev S.N., Lebedeva T.V., Najdenova S.V., Pisarev V.N., Pronina E.A., Rykovanova V.V., Selezneva K.I., Sizova I.E., Chernova S.V., Chegolev E.V. - “Calculations of Thermal Regime of the Muon Chambers Volume of ATLAS experiment” - EDMS 493977

[8] Anna Mueller - The 2D CFD study of the thermal behavior of the ATLAS Muon chambers and the cavern” - EDMS 592555

[9] Moritz Kuhn - “Air cooling of CNGS Horn structure efficiency study” - EDMS 492210

[10] Izabella Wichrowska-Polok - “Airflow in the CNGS Tunnel Structures in Case of Decay Tunnel Titanium Window Rupture” - EDMS 492187

[11] “Numerical Analysis Request Template” - EDMS 459266

[12] “Efficient Use of Computational Fluid Dynamics for the Numerical Solution of Thermo-Fluid Mechanics" - Centro Interdipartimentale di Fluidodinamica e Idraulica of Udine University (Italy) - EDMS 573622

[13] “Computational Fluid dynamics at CERN - Presentation on 9th June 2004” - EDMS 476523

[14] “Computational Fluid dynamics at CERN - Presentation on 15th April 2005” - EDMS 581908

[15] CFD Flyer: https://edms.cern.ch/file/476523/1/CFD_flyer.pdf

[16] “Statement of support of the LHCG Computing Coordinators for the resources allocation to the CFD team of Detector Cooling Section” - EDMS 556253