Computational Fluid Flow and Heat Transfer, or simply CFD, has become a well-accepted tool in the design of many products and processes in the industrial sector. It has also become an enabling technology for research in an increasing number of disciplines, spanning from the traditional engineering sector up to the life sciences. The number of available CFD methods and approaches is rather large, however, traditionally, students at MSc level have only been exposed to a few of these. Moreover, the use of these tools for research tasks, even when they are packaged in friendly commercial platforms, would benefit greatly from a more in-depth knowledge of their characteristics.
The 16th UIT Summer School attempts fill this gap by providing an intensive, one-week course focused on the fundamental methods used in Computational Fluid Flow and Heat Transfer. The lectures will span from traditional methodologies, like Finite Volume and Finite Element, to the more recent Lattice Botlzmann. Methods for multiphase flows will be also addressed, together with useful advice on how to use both commercial and open source software and an example of custom code development. Two CFD major vendors will illustrate the opportunities offered by modern CFD platforms with relevant examples and case studies.

CONTRIBUTORS

Diego Angeli, Università di Modena e Reggio Emilia
Pietro Asinari, Politecnico di Torino
Giulio Croce, Università di Udine
Stephane Cyr, EXA Corporation, Burlington, MA, USA
Norman Del Puppo, Sauber F1 Team, Hinwil, CH
Davide Frigerio, ANSYS Italia srl
Fabio Inzoli, Politecnico di Milano
Paolo Levoni, Università di Modena e Reggio Emilia
Oronzio Manca, Seconda Università di Napoli
Enrico Nobile, Università di Trieste
Carlo Nonino, Università di Udine
Pietro Poesio, Università di Brescia

ADDITIONAL INFORMATIONS

Additional info about the Summer Schools can be found on the UIT website: www.uitonline.it
For any further questions and requests, please contact: Prof. Enrico Nobile, Director of 16th UIT Summer School (nobile@units.it).

CREDITS FOR PHD STUDENTS

PhD Students can gain credits according to the regulation of their own PhD School. In addition to the Attendance Certificate, a Proficiency Certificate can be obtained upon submission of a report, to be agreed, on one of the topics addressed in the program of the School.

APPLICATION AND FEES

The registration fee is 700,00 Euros and includes attendance to the Summer School, coffee breaks during the lessons, and full board treatment from the dinner of Sunday, August 28th to the lunch of Friday, September 2nd. The 50% of registration fee (€ 350,00) must be paid before August 12, 2016. The remaining must be paid directly during the check-in at Certosa di Pontignano.
To apply, please download (www.uitonline.it) and complete the registration form, and kindly send it by e-mail, before August 12, 2016, to:
info@lacertosadipontignano.com and, in cc:, to:
nobile@units.it

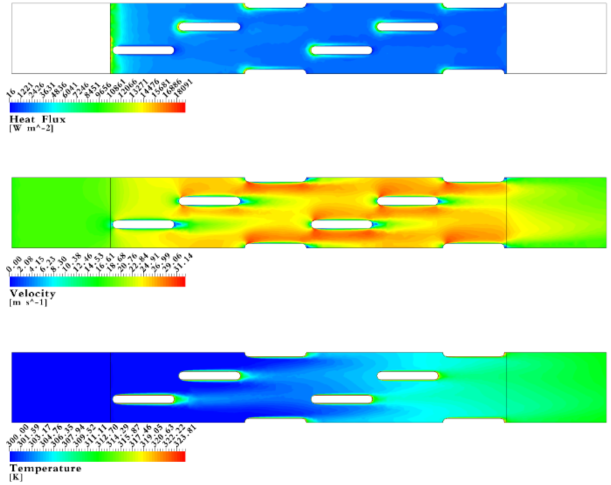
LOCATION

The 16th Summer School will be held in the prestigious Ancient Certosa di Pontignano, a unique place where nature, history and hospitality blend together in a memorable harmony, at a few kilometers from Siena, in the heart of Chianti, on a hill dominating the town. Further information can be gathered directly at Certosa website (www.lacertosadipontignano.com).

UIT Unione Italiana Termofluidodinamica

16th UIT Summer School

August 29 - September 2, 2016



COMPUTATIONAL FLUID FLOW AND HEAT TRANSFER

Director: Prof. Enrico Nobile – Università degli Studi di Trieste

Certosa di Pontignano (Siena)

CARTHUSIA PONTINIANI
SIENA

PROGRAMME

	Monday 29 August	Tuesday 30 August	Wednesday 31 August	Thursday 1 September	Friday 2 September
9:00	G. Croce *Fundamental equations for thermal-fluid problems*	C. Nonino *Finite Element Method*	D. Angeli, P. Levoni *Commercial CFD codes*	F. Inzoli *Turbulence/CFD management*	O. Manca *Custom code development – an example*
9:45	G. Croce *Fundamental equations for thermal-fluid problems*	C. Nonino *Finite Element Method*	D. Angeli, P. Levoni *Commercial CFD codes*	F. Inzoli *Turbulence/CFD management*	O. Manca *Custom code development – an example*
10:30	***Coffee break***	***Coffee break***	***Coffee break***	***Coffee break***	***Coffee break***
11:00	G. Croce *Fundamental equations for thermal-fluid problems*	C. Nonino *Finite Element Method*	D. Angeli, P. Levoni *Commercial CFD codes*	F. Inzoli *Turbulence/CFD management*	Closing meeting & Depar-tures
11:45	E. Nobile *Finite Volume Method*	P. Poesio *Methods for multiphase flows*	N. Del Puppo *CFD with open source software*	P. Asinari *Lattice Boltzmann Methods*	Closing meeting & Depar-tures
13:00	***Lunch***	***Lunch***	***Lunch***	***Lunch***	***Lunch****
14:30	E. Nobile *Finite Volume Method*	P. Poesio *Methods for multiphase flows*	N. Del Puppo *CFD with open source software*	P. Asinari *Lattice Boltzmann Methods*	Departures
15:15	E. Nobile *Finite Volume Method*	P. Poesio *Methods for multiphase flows*	N. Del Puppo *CFD with open source software*	P. Asinari *Lattice Boltzmann Methods*	
16:00	***Coffee break***	***Coffee break***	***Coffee break***	***Coffee break***	
16:30		CFD vendor presentation *EXA Corporation, USA*		CFD vendor presentation *ANSYS Italia*	
17:15		CFD vendor presentation *EXA Corporation, USA*		CFD vendor presentation *ANSYS Italia*	
18:00					
20:00	***Dinner***	***Dinner***	***Dinner***	***Dinner***	
21:00		***Social event:*** *Vin santo e cantucci*			

* For the lunch of Friday, September 2nd, each participant is kindly asked to confirm his/her presence at the *reception*.

LIST OF CONTRIBUTORS

- Diego Angeli — Università degli Studi di Modena e Reggio Emilia (diego.angeli@unimore.it)
- Pietro Asinari — Politecnico di Torino (pietro.asinari@polito.it)
- Giulio Croce — Università degli Studi di Udine (giulio.croce@uniud.it)
- Stephane Cyr — EXA Corporation, Burlington, MA, USA (stephane@exa.com)
- Norman Del Puppo — Sauber F1 Team, Hinwil, CH (normandelpuppo@gmail.com)
- Davide Frigerio — ANSYS Italia srl (davide.frigerio@ansys.com)
- Fabio Inzoli — Politecnico di Milano (fabio.inzoli@polimi.it)
- Paolo Levoni — Università degli Studi di Modena e Reggio Emilia & Mimesis s.r.l., Modena (paolo.levoni@unimore.it)
- Oronzio Manca — Seconda Università degli Studi di Napoli (oronzio.manca@unina2.it)
- Enrico Nobile — Università degli Studi di Trieste (nobile@units.it)
- Carlo Nonino — Università degli Studi di Udine (carlo.nonino@uniud.it)
- Pietro Poesio — Università degli Studi di Brescia (pietro.poesio@unibs.it)

CFD VENDORS

- ANSYS Italia srl (www.ansys.com)
- EXA Corporation, Burlington, MA, USA (www.exa.com)

LECTURE NOTES

Before the beginning of the Summer School, the slideshows and/or notes of the lectures will be available for download in a restricted access area of the UIT website (http://www.uitonline.eu).

LOCATION

The 16th Summer School will be held at Certosa di Pontignano (Siena); further information can be gathered directly at Certosa website.

APPLICATION AND FEES

The registration fee is 700,00 Euros and includes attendance to the Summer School, coffee breaks during the lessons, and full board treatment from the dinner of Sunday, August 28th to the lunch of Friday, September 2. Each participant is kindly asked to confirm at the reception his/her presence at the lunch of Friday, September 2.
The 50% of registration fee (€ 350,00) must be paid before August 12, 2016, following the instructions given within the attached registration form. The remaining 50% (€ 350,00) must be paid directly during the check-in at Certosa di Pontignano.
Please, to apply download and complete (in PDF or RTF format) the registration form, and kindly send it by e-mail, before August 14, 2016, to: info@lacertosadipontignano.com.

CREDITS FOR PHD STUDENTS

PhD Students can gain credits according to the regulation of their own PhD School. In addition to the Attendance Certificate, a Proficiency Certificate can be obtained upon submission of a report , to be agreed, on one of the topics addressed in the pro-gram of the School.
Additional info about the Summer Schools are available on www.uitonline.eu.

For any further questions and requests, please contact:
Prof. Enrico Nobile, Director of the 16th UIT Summer School (nobile@units.it)
Prof. Giulio Croce, Vice-Director of the 16th UIT Summer School (giulio.croce@uniud.it)
Prof. Sara Rainieri, Secretary of UIT Steering Committee (sara.rainieri@unipr.it)

SUMMARIES

Giulio Croce

Fundamental equations for thermal-fluid problems

The lecture offers an overview of the fundamental equations for thermo fluid dynamics phenomena. The focus is on the physical and mathematical characteristics that may affect the numerical discretization: thus, after deriving the continuity, momentum and energy equations for compressible and incompressible flows from local balances, care is given to the evaluation of wave propagation speed and direction, their effect on the physical boundary conditions definition, and their impact on the numerical modelling. Uncompressible and compressible flow equation are considered, including comments on the discontinuities in compressible flow solution. Finally, similar form of transport equations will be briefly discussed (passive scalar transport, shallow water, mass transfer, mixing), as well as possible limitation of the standard Navier-Stokes equations, such as those arising in micro-scale applications.

Enrico Nobile

Finite Volume Method

The Finite Volume (FV) method gained popularity among CFD practitioners, using either commercial or open source CFD platforms, and in the academic community at large. The reason of this success is due to several aspects, which are peculiar of this method. Its simple physical interpretation, which leads to the use of the FV as the method of choice for introducing students to CFD. Its strict conservative property, due to the transformation of some of the terms of the original conservation equations in face fluxes. Finally its high geometrical flexibility, obtained by the use of grids constituted by arbitrary polyhedra. These are probably the most important factors. In this lecture, the FV method will be introduced considering first Cartesian grids, which lead automatically to simple data structures and easy coding, followed by unstructured grids, which require more complex data structures but provide, as a reward, the possibility to describe complex, and possibly time-varying, geometries.

Carlo Nonino

Finite Element Method

The Finite Element method (FEM) is a numerical technique for the solution of partial differential equations and is the dominant discretization technique for structural analysis. The basic concept in the physical interpretation of the FEM is the subdivision of the computational domain into disjoint (non-overlapping) components of simple geometry, which are called finite elements and form a mesh. After the first applications in the field of solid mechanics in the early 1960s, the FEM was soon extended to the solution of heat transfer and fluid flow problems. No conservation principles are explicitly enforced, but the method has strong theoretical bases. Its great flexibility, together with the possibility of using unstructured meshes make the FEM a very convenient method for the solution of many multiphysics problems (e.g.: fluid-structure interaction). In this lecture, a description of how the FEM works will be presented with reference to the solution of 1-D and 2-D steady-state and transient heat transfer problems.

Pietro Poesio

Methods for multiphase flows

The lecture provides an overview of the models and numerical methods to simulate two-phase flows, focusing, in particular, on the Direct Numerical Simulations approach. The most peculiar aspect of multiphase flows is the presence of an interface whose effects need to be accounted for. In DNS the interface is described explicitly by the kinematic boundary condition, which is represented by an advection equation. Different ways to solve and represent the equation give rise to different modeling approaches. In the first part of lecture we will focus on slug capturing. Broadly specking, within the slug capturing framework one can use the Volume of Fluid or the Level Set approach. The second part of the lecture will introduce and describe the more recent slug tracking approach. In the last part of the lecture, we will focus on the Eulerian-Eulerian approach, where the interface is not represented in an explicitly way, but by averaged equations where source terms are introduced.

Paolo Levoni and Diego Angeli

Commercial CFD codes

This lecture aims at introducing commercial CFD codes as a tool for both scientific research and academic development. The growth of commercially-available CFD software in the last 25 years has been tremendous, and nowadays most codes are virtually capable of dealing with multiple physical phenomena, coupled with the modeling of turbulent flow. More than ever, it is critical to understand the complexity and variety of such tools in order to perform reliable and physically sound analyses. In this lecture, the main potentialities, limits, pitfalls and risks of commercial CFD will be illustrated, starting from the study of the physical problem, and going through the definition of the mathematical model, the meshing process, the implementation of boundary and initial conditions, the numerical solution, the post-processing of results, and verification checks on the reliability of results. Finally, some examples of the applications of commercial CFD for the solution of thermo-fluid dynamics problems in research will be presented.

Norman Del Puppo

CFD with open source software

Today, numerical simulations are indispensable in industrial production. Many industries uses Computational Fluid Dynamics (CFD) to predict the flow behavior inside or outside a product in order to understand better the various characteristics of any product design. Numerical investigations dramatically reduce the cost of designing new products for which fluid dynamics is a large part of the value-added. Nowadays, the high cost of the software is one, among the others, of the main drawbacks that more affects a widespread use of this theoretical approach to the design of new products. Recently, Open Source CFD software packages has become popular among universities and industries mainly because of the absence of a license cost and the possibility to customize the software to the user needs. The purpose of this talk is to give an overview of OpenFOAM® (Weller, Tabor, Jasak, & Fureby, 1998), an open source library based on the Finite Volume Method (FVM), which has gained popularity in the CFD community over the last decade. There will be given an introduction in the use of OpenFOAM® for computational fluid dynamics analysis and to the library code organization. Finally, there will presented some illustrative examples of custom software developments applied to marine and motorsport applications.

Fabio Inzoli

Turbulence Modelling

Many flows of engineering interest are turbulent. Turbulence causes the appearance in the flow of eddies with a wide range of length and time scales that interact in a dynamically complex way.
The lecture gives a brief introduction to the physics of turbulence and to its modelling in CFD. For most engineering purposes it is unnecessary to resolve the details of the turbulent fluctuations. CFD analysts are almost always satisfied with information about the time-averaged properties of the flow (e.g. mean velocities, mean pressures, mean stresses etc.). Therefore, the vast majority of turbulent flow computations has been and for the foreseeable future will continue to be carried out with procedures based on the Reynolds-averaged Navier–Stokes (RANS) equations. The most widely used turbulence models (k-ε, k-ω, RSM, etc..) are discussed.
A brief introduction to LES modelling is presented.

CFD management

Computational Fluid Dynamics analysis has become a useful research and design instrument albeit with errors and uncertainties. Appropriate standards and protocols for increasing confidence and reliability need to be identified and applied. These requirements become more relevant as improvements in hardware, software and user competence increase. Advances in these sectors have led to the increased use of CFD codes in the applied research industry.
In the discussion on the opportunities of applying Quality Assurance to Research and Development activities, and on investigating methodologies (numerical or experimental), CFD results need to be qualified; rules, and procedures should be instated and followed.
The lecture aims to focus on methodological approach to qualify CFD. The approach is based on three interdependent, but related, dimensions: software reliability, user knowledge and process control. Applying the quality of CFD analysis to industrial problems, while following the principles of Quality Assurance, is of particular concerned.

Pietro Asinari

Lattice Boltzmann Methods

This lecture aims to introduce the basic concepts of the Lattice Boltzmann Method (LBM), which defines a class of computational fluid dynamics (CFD) methods for fluid simulation. This approach appears as well-suited to high performance computing (HPC) and in particular to massively parallel processors such as graphics processing units (GPUs), which are expected to become more and more popular in the future. The lecture starts with the historical kinetic interpretation of the method, but then it aims to focus on its essential peculiarities, which make it computationally effective for solving continuum equations. The mathematical analysis of the method is illustrated by a simple example based on the conduction equation. An overview of some relevant engineering applications is presented. Moreover the basic ingredients of the computational algorithm are discussed by means of some hands-on example codes, which will be distributed to the audience.

Oronzio Manca

Custom code development – an example

The lecture presents a development of a numerical procedure related to natural convection in channel. It is just an example to introduce and encourage graduate students and young researchers to try to build their own code with numerical thermofluidynamics knowledge. In the first part an introduction of the physical problem and the governing equations in dimensional and non-.dimensional form is given together with a description of the main dimensionless numbers in natural convection. The solution by boundary layer theory is described and the main limits of this approach are presented. The problem of the boundary condition in natural convection in open channels is highlighted and some choices of different computational domains are discussed. Stream function-vorticity formulation is presented together with the boundary conditions. The numerical analysis and the numerical algorithm of stream function - vorticity formulation is given and the problems related to the evaluation of pressure field in this formulation are underlined. The extension of the formulation to the transient regime is presented. The verification and validation processes are explained and their application in the considered problem is accomplished. Some discussion on the results are given to complete the topic.