Presentation of the courses:

Introduction to Computational Fluid Dynamics
Computational methods for Fluid Dynamics and Heat Transfer

Enrico Nobile

Dipartimento di Ingegneria e Architettura
Università degli Studi Trieste, 34127 TRIESTE

3 marzo 2025

OUTLINE

Objectives

- Provide the theoretical and practical elements to allow an informed use of *Computational Thermofluid Dynamics* or *TCFD* (*CFD - Computational Fluid Dynamics*) techniques in the applied and industrial field;
- To verify the ways, and the related problems, with which CFD techniques can be integrated into the *Design Chain*;
- Reconcile the two aspects of CFD – theoretical foundations and application methods – which are often treated separately;
- Provide *food for thought* on the current situation and future prospects of CFD in the field of CAE (Computer Aided Engineering) and more generally of *virtual prototyping* [1]
- To outline the possibility to integrate and/or to embed CFD models into the Digital Twin (DT) concept.

[1] from Wikipedia: *Virtual prototyping is a technique in the process of product development. It involves using computer-aided design (CAD), computer-automated design (CAutoD) and computer-aided engineering (CAE) software to validate a design before committing to making a physical prototype. This is done by creating (usually 3D) computer generated geometrical shapes (parts) and either combining them into an "assembly" and testing different mechanical motions, fit and function or just aesthetic appeal. The assembly or individual parts could be opened in CAE software to simulate the behavior of the product in the real world.*

Objectives - *cont.*

Specifically, what will be the skills and abilities acquired at the end of the course?

- Know, with a good level of detail, the spatial and temporal discretization techniques most used in commercial (industrial) and *Open Source* CFD packages;
- Recognize and be able to judge the correctness of the general setup of a CFD analysis;
- Understand the typical workflow of a modern CFD package;
- Know how to use, for problems of low geometric and physical complexity - as well as computational - a modern *commercial* and partly *Open Source* - CFD package;
- Be able to tackle and solve low/medium difficulty thermo-fluid dynamic problems using CFD tools (ANSYS Fluent and, to a lesser extent, ANSYS CFX);
- Know how to use a commercial mesh generator (ANSYS Mesh, Fluent Mesh and/or ANSYS ICEM CFD) in problems of low geometric complexity;
- Know how to use MATLAB, and the PDE Toolbox (Partial Differential Equation Toolbox), to develop scripts/functions that implement what has been learned in class.

Potential interested parties

- Students of Master's Degree courses in Engineering:
  - **Mechanical**
  - **Engineering for the Energy Transition**
  - Naval architecture
  - Materials and Chemical Engineering for Nano, Bio, and Sustainable Technologies
  - Electrical Energy and Systems Engineering
  - Other courses...
- Students of the Master's Degree courses in Mathematical, Physical and Natural Sciences:
  - Physics
  - Mathematics
  - Scientific and Data-Intensive Computing
  - Data Science And Artificial Intelligence
- Industrial Engineering PhD students – but this is NOT a specific course for PhD students!

Necessary and useful skills

- Core courses[2]
  - *V.O*: Analisi I e II, Fisica Generale I, Fisica Tecnica, Fluidodinamica o Idraulica (recommended);
  - *L.T.* + corso base Termodinamica e Trasmissione del calore (Fisica Tecnica I o Fisica Tecnica) e Fluidodinamica o Idraulica (recommended) ;
- Numerical methods, programming, IT tools skills:
  - Fundamentals of *Numerical Analysis*: very useful but not indispensable.
  - Any language, preferably *MATLAB*®, *Python* or *Julia*: extremely useful, although not strictly essential. The use of compiled languages (*C*, *C++*, *Fortran*) is possible but not recommended.
  - Computer tools (LaTeX, vector graphics programs, etc.): useful but not essential.
  - Knowledge of 3D CAD systems and solid modeling fundamentals: useful but not essential.

[2] Below are the most common names of the courses offered at Italian Universities, but similar courses, in terms of content, held at foreign universities are also valid.

Course contents - summary

- Introduction to Computational Fluid Dynamics - CFD and Numerical Heat Transfer - NHT - Overview
  - Whats is CFD?
  - Applications of CFD
  - Applications and (present) limitations of CFD
- Historical notes
- Examples of application
- Introduction to conservation laws:
  - Mass, energy and momentum conservation equations;
  - Solution in terms of primitive variables and stream function-vorticity;
  - Algorithms for incompressible flows.
- *Dimensional Analysis and Similitude*[3];

[3] Independent learning - flipping class

Presentation of the courses: Intro to CFD & Comp. meth. for Fluid Dyn. Heat Tr. E. Nobile | Course program

Course contents - summary- *cont.*

- Numerical methods for CFD and NHT:
  - Components of CFD: mathematical model, domain, grid et.
  - Properties of numerical methods
  - Stability
  - Discrete approaches: Finite Difference (FD), Finite Volume (FV), Finite Element (FE), some notes on other methods
- Fundamentals of Finite Difference method (FD);
- The Finite Volume method (FV) for incompressible flows:
  - Cartesian/structured grids
  - Unstructured grids
- The Finite Element method (FE) for incompressible flows;
- Turbulence and its models:
  - DNS - Direct Numerical Simulation
  - LES - Large Eddy Simulation
  - RANS - Reynolds-Averaged Navier-Stokes equations
  - DES/DDES ((Delayed) Detached Eddy simulation) - few notes
- Verification & Validation (V & V);
- Accuracy and convergence.

Texts and teaching materials

1. G. Comini, G. Croce, E. Nobile, FONDAMENTI DI TERMOFLUIDODINAMICA COMPUTAZIONALE (*in Italian*), SGEditoriali, 4° Edizione, Padova, 2014. Now no longer available, digital copies of the chapters of interest will be provided.
2. Notes, transparencies, and tutorial texts distributed in electronic format on *MOODLE FEDERATO* *480MI-2 - COMPUTATIONAL METHODS FOR FLUID DYNAMICS AND HEAT TRANSFER 2024*
3. Wide availability of texts and articles for further study at the *Area biblioteche di scienze, tecnologie e scienze della vita* (Via Valerio 10, Building C1, first floor)

CFD books

Non-exhaustive list of CFD books

General

- S.V. Patankar, *NUMERICAL HEAT TRANSFER AND FLUID FLOW*, Hemisphere Publishing Corporation, https://doi.org/10.1201/9781482234213, 1980.
- J. D. Anderson, *COMPUTATIONAL FLUID DYNAMICS - THE BASICS WITH APPLICATIONS*, McGraw-Hill, 1995.
- D. Anderson, J. C. Tannehill, R. H. Pletcher, R. Munipalli, V. Shankar, *COMPUTATIONAL FLUID MECHANICS AND HEAT TRANSFER*, CRC Press, 4th Ed. 2020.
- J H. Ferziger, M. Perić, *COMPUTATIONAL METHODS FOR FLUID DYNAMICS*, 4th ed., Springer Berlin, https://doi.org/10.1007/978-3-319-99693-6, 2019.
- H. Lomax , Thomas H. Pulliam , David W. Zingg, *FUNDAMENTALS OF COMPUTATIONAL FLUID DYNAMICS*, Springer-Verlag Berlin, https://doi.org/10.1007/978-3-662-04654-8, 2011
- T. J. Chung, *COMPUTATIONAL FLUID DYNAMICS*, 2nd ed., Cambridge University Press, https://doi.org/10.1017/CBO9780511780066, 2010.
- J. Blazek, *COMPUTATIONAL FLUID DYNAMICS: PRINCIPLES AND APPLICATIONS*, 3rd Ed., Elsevier, https://doi.org/10.1016/B978-0-08-099995-1.09985-1, 2015

CFD books - *cont.*

FV

- H.K. Versteeg and W. Malalasekera, *AN INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS: THE FINITE VOLUME METHOD*, 2nd ed., Addison-Wesley, 2007.
- F. Moukalled, L. Mangani and M. Darwish, *THE FINITE VOLUME METHOD IN COMPUTATIONAL FLUID DYNAMICS - AN ADVANCED INTRODUCTION WITH OpenFOAM® AND MATLAB®*, https://doi.org/10.1007/978-3-319-16874-6, 2016.
- Clovis R. Maliska, *FUNDAMENTALS OF COMPUTATIONAL FLUID DYNAMICS - THE FINITE VOLUME METHOD*, https://doi.org/10.1007/978-3-031-18235-8, 2023.

FEM

- O.C. Zienkiewicz, R.L. Taylor and P. Nithiarasu, *THE FINITE ELEMENT METHOD FOR FLUID DYNAMICS*, 7th Ed., Elsevier, https://doi.org/10.1016/C2009-0-26328-8, 2014.

Spectral method

- G. Karniadakis, S. Sherwin, *SPECTRAL/HP ELEMENT METHODS FOR COMPUTATIONAL FLUID DYNAMICS: SECOND EDITION*, Oxford University Press, 2004.

Lattice Boltzmann method

- S. Succi, Sauro, *THE LATTICE BOLTZMANN EQUATION FOR FLUID DYNAMICS AND BEYOND*, Oxford University Press, 2001.
- T. Krüger, H. Kusumaatmaja, A. Kuzmin, O. Shardt, G. Silva, E. M. Viggen, *THE LATTICE BOLTZMANN METHOD: PRINCIPLES AND PRACTICE*, Springer Verlag, https://doi.org/10.1007/978-3-319-44649-3, 2016.

Some pages from the text

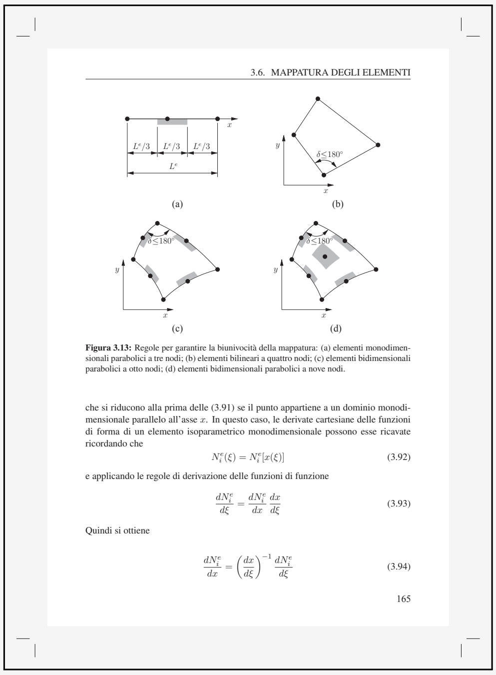

3.6. MAPPATURA DEGLI ELEMENTI

Figura 3.13: Regole per garantire la biunivocità della mappatura: (a) elementi monodimensionali parabolici a tre nodi; (b) elementi bilineari a quattro nodi; (c) elementi bidimensionali parabolici a otto nodi; (d) elementi bidimensionali parabolici a nove nodi.

che si riducono alla prima delle (3.91) se il punto appartiene a un dominio monodimensionale parallelo all'asse x. In questo caso, le derivate cartesiane delle funzioni di forma di un elemento isoparametrico monodimensionale possono esse ricavate ricordando che

$$N_i^e(\xi) = N_i^e[x(\xi)] \quad (3.92)$$

e applicando le regole di derivazione delle funzioni di funzione

$$\frac{dN_i^e}{d\xi} = \frac{dN_i^e}{dx}\frac{dx}{d\xi} \quad (3.93)$$

Quindi si ottiene

$$\frac{dN_i^e}{dx} = \left(\frac{dx}{d\xi}\right)^{-1}\frac{dN_i^e}{d\xi} \quad (3.94)$$

165

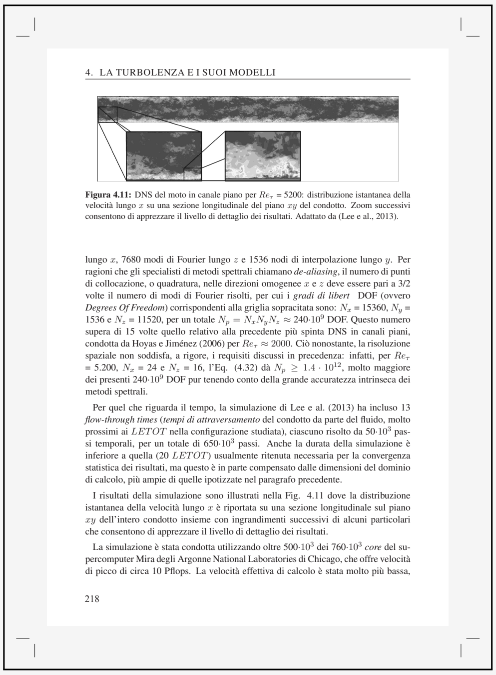

4. LA TURBOLENZA E I SUOI MODELLI

Figura 4.11: DNS del moto in canale piano per Re_τ = 5200: distribuzione istantanea della velocità lungo x su una sezione longitudinale del piano xy del condotto. Zoom successivi consentono di apprezzare il livello di dettaglio dei risultati. Adattato da (Lee e al., 2013).

lungo x, 7680 modi di Fourier lungo z e 1536 nodi di interpolazione lungo y. Per ragioni che gli specialisti di metodi spettrali chiamano *de-aliasing*, il numero di punti di collocazione, o quadratura, nelle direzioni omogenee x e z deve essere pari a 3/2 volte il numero di modi di Fourier risolti, per cui i *gradi di libert* DOF (ovvero *Degrees Of Freedom*) corrispondenti alla griglia sopracitata sono: N_x = 15360, N_y = 1536 e N_z = 11520, per un totale $N_p = N_x N_y N_z \approx 240 \cdot 10^9$ DOF. Questo numero supera di 15 volte quello relativo alla precedente più spinta DNS in canali piani, condotta da Hoyas e Jiménez (2006) per $Re_\tau \approx 2000$. Ciò nonostante, la risoluzione spaziale non soddisfa, a rigore, i requisiti discussi in precedenza: infatti, per Re_τ = 5.200, N_x = 24 e N_z = 16, l'Eq. (4.32) dà $N_p \geq 1.4 \cdot 10^{12}$, molto maggiore dei presenti $240 \cdot 10^9$ DOF pur tenendo conto della grande accuratezza intrinseca dei metodi spettrali.

Per quel che riguarda il tempo, la simulazione di Lee e al. (2013) ha incluso 13 *flow-through times* (*tempi di attraversamento* del condotto da parte del fluido, molto prossimi ai $LETOT$ nella configurazione studiata), ciascuno risolto da $50 \cdot 10^3$ passi temporali, per un totale di $650 \cdot 10^3$ passi. Anche la durata della simulazione è inferiore a quella (20 $LETOT$) usualmente ritenuta necessaria per la convergenza statistica dei risultati, ma questo è in parte compensato dalle dimensioni del dominio di calcolo, più ampie di quelle ipotizzate nel paragrafo precedente.

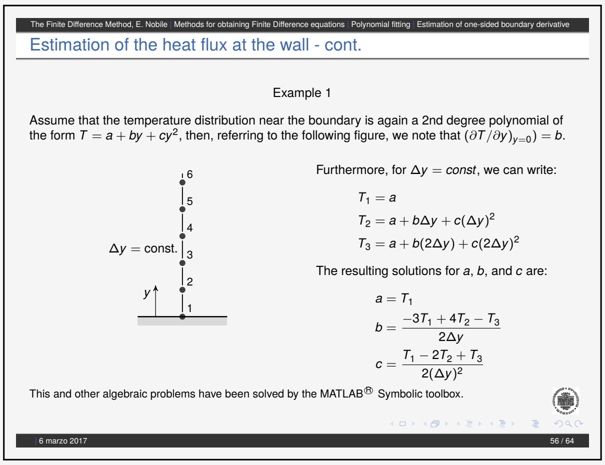
I risultati della simulazione sono illustrati nella Fig. 4.11 dove la distribuzione istantanea della velocità lungo x è riportata su una sezione longitudinale sul piano xy dell'intero condotto insieme con ingrandimenti successivi di alcuni particolari che consentono di apprezzare il livello di dettaglio dei risultati.

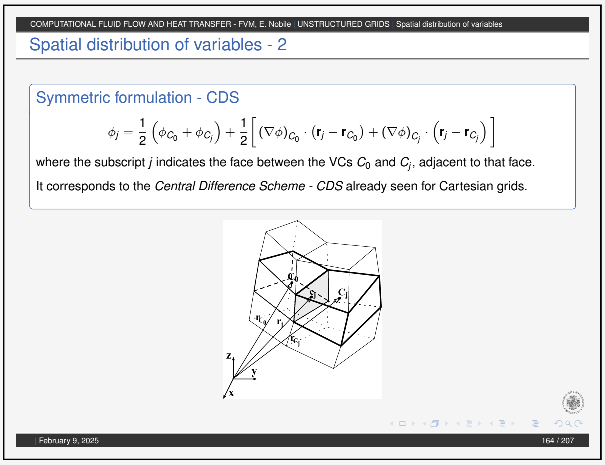
La simulazione è stata condotta utilizzando oltre $500 \cdot 10^3$ dei $760 \cdot 10^3$ *core* del supercomputer Mira degli Argonne National Laboratories di Chicago, che offre velocità di picco di circa 10 Pflops. La velocità effettiva di calcolo è stata molto più bassa,

218

Some pages from the text *FONDAMENTI DI TERMOFLUIDODINAMICA COMPUTAZIONALE, 4th Ed.*.

Slide examples





Examples of slides used in lectures and available online.

Communication methods

- For several years, to report updates to the online teaching material, possible changes to classrooms, upcoming seminars and other communications, the *Forum* of the course on *MOODLE* has been used *exclusively*.
- Students will still be able to contact the teacher(s) directly via email.
- *Moodle* page for the course(s):

    `https://moodle2.units.it/course/view.php?id=15138`

Tutorials - characteristics and purposes

- The tutorials will consist of the step-by-step solution, with the support of the teachers, of some problems, from the simplest ones to cases of industrial interest, which will fully illustrate the correct way of using a commercial (industrial) CFD package.
- The problems will be selected, based on needs, from the following:
  1. Calculation of heat transfer from a fin (MATLAB)
  2. Calculation of thermal bridges according to *EN ISO 10211: Thermal bridges in building construction* (MATLAB/PDEToolbox - Fluent)
  3. Turbulent flow in a 90° elbow (CFX/Fluent)
  4. Laminar and turbulent flow around a cylinder (Fluent)
  5. Block-structured meshing for the flow around a cylinder (ANSYS mesh)
  6. Prediction of the thermal performance of a microchannel heat sink (Fluent)
  7. Multiphase flow of a water column - *breaking dam* (CFX/Fluent)
  8. Analysis of a simple centrifugal pump (CFX)
  9. External aerodynamics over Ahmed's body (Fluent)
  10. Gallium melting in a rectangular cavity heated from the side (Fluent)
  11. Meshing of a static mixer (Fluent).
- The aim is to allow the student to become familiar with modern CFD tools aimed at CAE applications and, more generally, at a variety of thermo-fluid dynamic problems.

Tutorials: practical aspects

- The CFD package chosen is a commercial (industrial) one, i.e. *ANSYS CFD®*, consisting of two distinct *general purpose* solvers:
  1. *ANSYS Fluent®*: general-purpose; continuously developed.
  2. *ANSYS CFX®*: historical reasons; focused on turbomachinery.
- Transition, for the 2024-25 Academic Year, to the ANSYS 2024 R1 (or ANSYS 2025 R1) release, in order to eliminate and/or mitigate any compatibility problems with the most recent operating systems, high-performance graphics cards, etc.
- As of August 17, 2015, the ANSYS® Student Portal is no longer available (see below), but *Ansys for Students* is available (`https://www.ansys.com/academic/students`).
- The hybrid and/or structured meshes required to run the simulations will be generated using the commercial meshing tools ANSYS® Mesh or, where applicable, ANSYS-ICEM® CFD.
- The tutorials will be carried out *in person*, in a suitable classroom equipped, in addition to WI-FI connection, also with electrical outlets for the students' laptops:
  - The possibility of attending lessons and tutorials *online* remains guaranteed for students who are eligible.
  - The lessons and tutorials will be recorded on the Microsoft Teams platform.

Software installation

- To allow the use of the package in different places and at different times, students who have their own Laptop - Windows, Linux or Mac - will be able to install the product on their machine, with the obvious advantage of being able to use it in other places.
- In order to use the program at home, you need to install a *Virtual Private Network* (VPN, detailed instructions available), to be able to access the *License Servers*:
  - Limited number – 100 – of *ANSYS Academic Multiphysics Campus Solution* licenses (as of November 2022) available;
  - Licenses limited to 4-16 cores.
- it is important, once the activity is finished, to correctly close ANSYS and its applications, so as to avoid keeping a license locked

ANSYS® Student

- At the same time as the ANSYS® Student Portal was discontinued, the ANSYS® products (with some limitations) were made available free of charge to students.
- In addition, ANSYS® provides students with access to self-study materials at `https://www.ansys.com/academic`, which include::
  1. Guides and videos for installing and configuring products.
  2. FAQ (PDF download).
  3. Introductory videos on simulation modes and *How To* videos.
  4. Links to academic sites that make available curricular materials and tutorials.
- Attention: ANSYS® student products obtained in this way CANNOT be used in class, nor even for carrying out the activities included in the course, for which it is instead necessary to use the educational licenses indicated by the teacher.
- Note: The ANSYS Student license is also characterized by limitations - available modules, max. cells/elements, etc. - not present for the Campus license available at the University.

Student project: characteristics

- *Student project* or *tesina*;
- Complete analysis and, in general, *validation*, through the ANSYS® CFD package (preferably Fluent or, for specific cases, CFX) plus possibly ANSYS ICEM CFD®, of a fluid dynamics problem:
  - The chosen problem, in the interest of the student, must be agreed with the teacher;
  - The choice of the problem to be analyzed may also be suggested by other professors, but must in any case be agreed with the teacher;
  - the problem can (should) be proposed by the student or, in case of difficulty, can be suggested by the teacher;
  - If the problem is chosen by the student, it is suggested to identify one or (better) more problems, in order to facilitate the selection by the teacher (see below).
- Alternatively, *autonomous* development, through a language for rapid prototyping (MATLAB®, Python, Julia, etc.) or generic, a simple code/application aimed at a specific problem.

Student project: characteristics of the problem

In general, unless otherwise agreed with the teacher, the aim of the *Student project* is to conduct a *validation* of the proposed numerical model through comparison with experimental results reported by other authors.
For this purpose the problem should be chosen based on the following considerations:

- Identify cases/problems for which accurate experimental findings are available and, where available, calculation results from other authors, in order to proceed with a critical comparison between CFD results and experimental measurements and, if possible, between current CFD results and CFD results from other authors;
- Prefer, in general, experimental data and discard *only* numerical results, so as to privilege the CFD-experimental comparison over the CFD-CFD or CFD-analytical one;
- Give priority to the availability of point data in addition to global quantities (e.g. availability of velocity profiles in addition to the *drag coefficient*; temperature values ??and profiles in addition to the average *Nusselt* number, etc.);
- Choose the source appropriately (ISI journal, report from a qualified institution, etc.), discard the so-called *predatory journals* see below;
- Discard problems of high geometric complexity (CAD file preparation and repair, meshing difficulties, etc.) which generally require more time to set up;
- Do not neglect more dated literature: simpler geometries, sometimes greater care and reliability of the data, etc.

Student project: choice of the problem I

A (non-exhaustive) list of sources for problem selection for the *Student project*:

Link	Comments
`https://turbmodels.larc.nasa.gov/`	■ Documentation related to RANS (Reynolds-averaged Navier-Stokes) turbulence models. ■ For validation see *Turbulence Model Validation Cases and Grids*.
`https://www.grc.nasa.gov/WWW/wind/valid/archive.html`	■ NASA NPARC Alliance Verification and Validation Archive. ■ Mostly compressible.
`http://cfd.mace.manchester.ac.uk/ercoftac/doku.php`	■ ERCOFTAC *Classic Collection* Database. ■ Possibility of search based on flow type.
`http://www.rpmturbo.com/testcases/index.html`	■ RPMTurbo is an engineering consultancy firm. ■ List of test cases for unsteady flow in turbomachinery. Steady-flow solutions are included.

Student project: choice of the problem II

Link	Comments
`https://www.aij.or.jp/jpn/publish/cfdguide/index_e.htm`	■ Guidebook for CFD Predictions of Urban Wind Environment. ■ Cases of interest for Civil Engineering.
`https://thtlab.jp/DNS/dns_database.html`	■ Turbulence and Heat Transfer Laboratory (THTLAB), University of Tokyo. ■ DNS Database of Turbulence and Heat Transfer for simple (fundamental) geometries.
`https://zcfd.zenotech.com/validation`	■ Private CFD consultancy firm. ■ Some validation cases, from simple ones (i.e. zero pressure gradient flat plate), to more complex cases (i.e. X15 at Mach number of 0.8).

Student project: choice of the problem III

Link	Comments
https://www.sciencedirect.com/	■ *Elsevier* platform that includes several Engineering and CFD scientific journals, e.g. *Computers & Fluids, Exp. Thermal and Fluid Science, Int. J. Heat and Mass Transfer, etc.* ■ Access through the University network or via VPN. Some *Open Access* articles are also available.

Student project: rules and suggestions

- Rules and suggestions for the Student project available in Moodle, and updated - almost - every Academic year;
- The *Student project* must be carried out in groups of 2(min)-4(Max) students:
  - The possibility of carrying out the Student project alone is possible only in exceptional cases and must be agreed with the teacher.
- Plagiarism check with Turnitin (`https://www.turnitin.com/`): check for similarity in the Student Project report.
- It must be sent, in *PDF* format, at least ten days before the oral exam, and a printed copy must be delivered (only once) on that occasion:
  - An initial draft should be sent for review. Then, there will be a one time opportunity to improve the project report based on my comments and suggestions.
  - The Student project evaluation constitutes an integral part of the exam grade;
  - Availability of (very limited) support in the development of the numerical/mathematical model;
  - An example of a *Student Project* prepared by the teachers is available.
  - A LaTeX *template* is available.

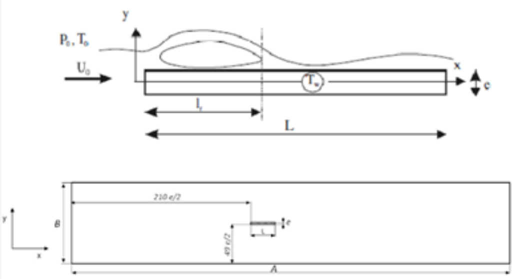
Student project: purpose

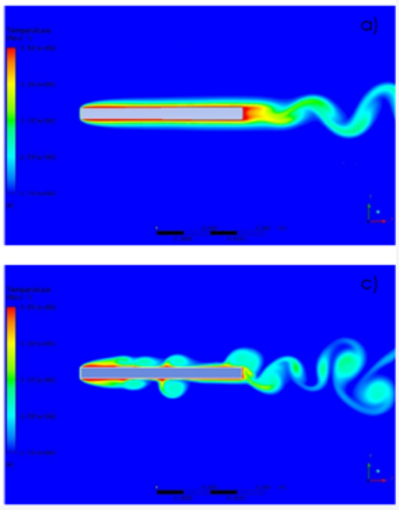
- Demonstrate the ability to tackle a problem independently, using advanced simulation methodologies;
- Develop *teamwork* skills;;
- Develop *problem solving* skills; ;
- Verify *hands-on* potential and limits of CFD;
- Learn a methodology as rigorous as possible for the use of CFD techniques, but of general validity for other CAE (Computer Aided Engineering) applications, for example structural, static and dynamic analysis, fatigue resistance, electromagnetic calculation etc.

Heat transfer for a flat plate of finite length and thickness

D. Vatta – AA. 2007-08

Scambio termico su lastra piana
di spessore finito



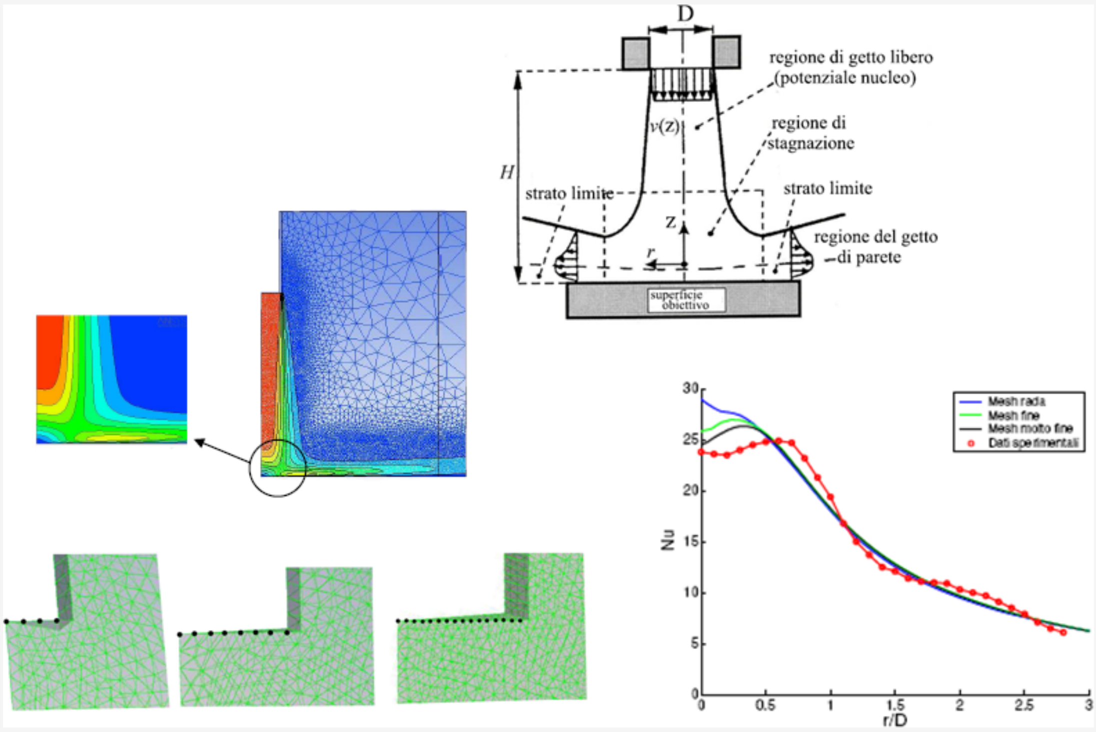


$Re_{e/2}$	$Nu_{e/2}$ CFX	DNS	Sperimentale	Errore % DNS/sperimentale	CFX/DNS	CFX/Sperimentale
120	1,02	1,07	1,20	10,80	4,70	15,00
360	2,57	3,14	3,00	4,50	18,20	14,30

Tabella 4-1 Confronto tra i dati ottenuti e quelli sperimentali e numerici di riferimento.

Heat transfer for an impinging jet

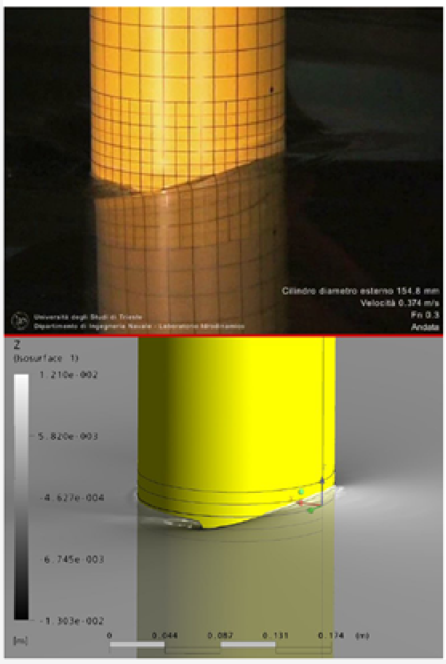
C. Curti, K. Meneguz – AA. 2005-06



Free surface around a cylinder

L. Ceolin, F. Fucile, D. H. Genuzio, M. Sidari – AA. 2005-06

Superficie libera attorno ad un corpo cilindrico; simulazione numerica e confronto sperimentale



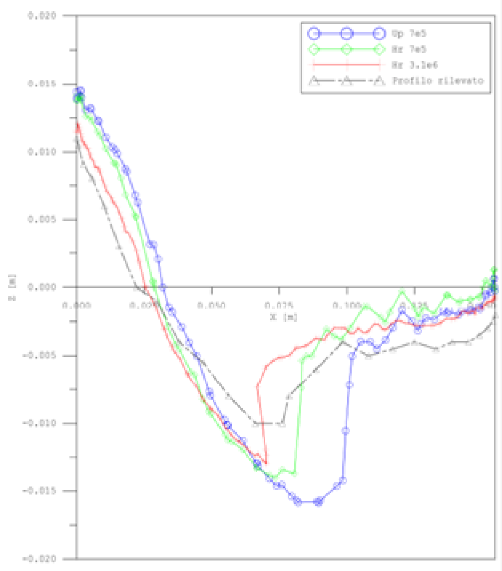
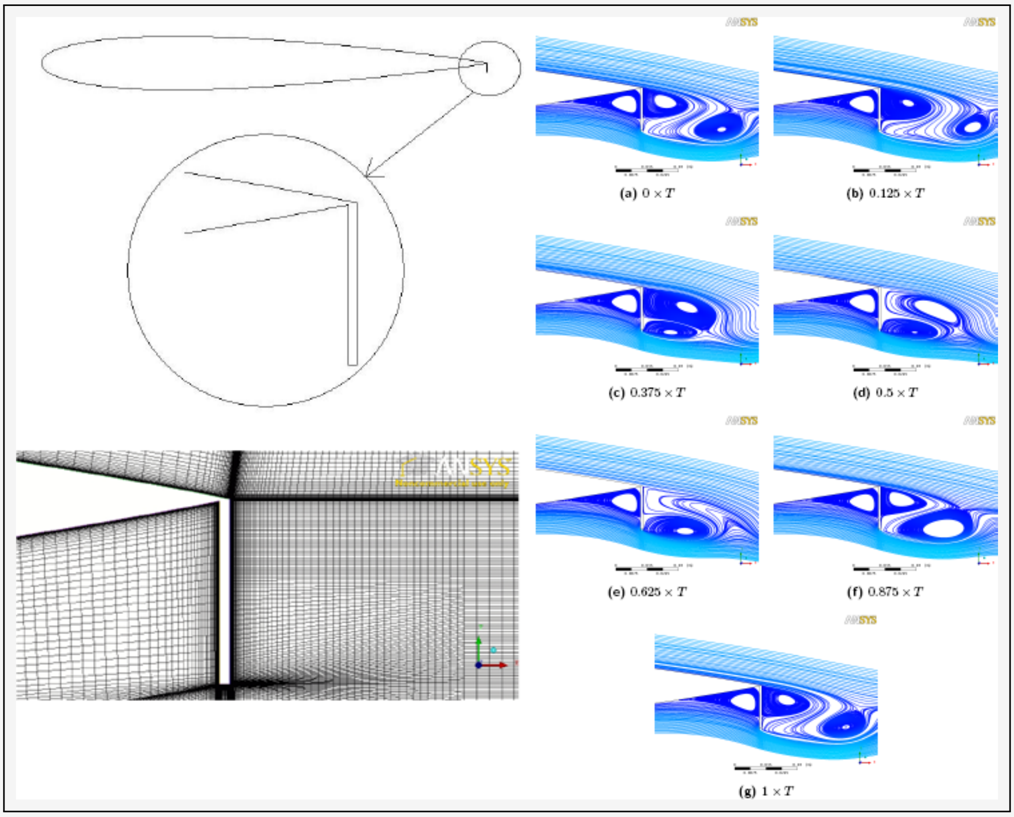


Figura 13: Diagramma di confronto elevazione della superficie libera

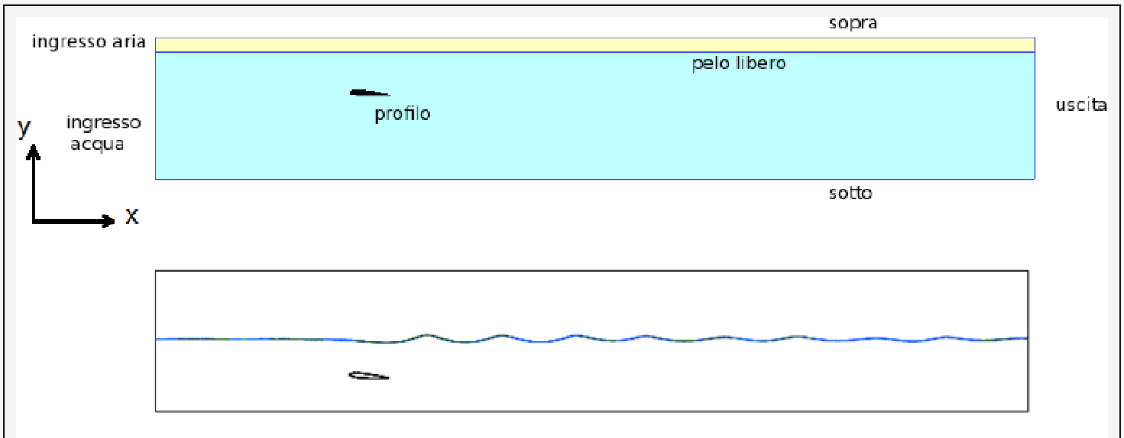
Gurney flap

M. Almerigogna, A. Scheri – AA. 2007-08



Hydrofoil

R. Boico, G. Stipcich – AA. 2009-10



(c) $p = 193\,mm$, $i = 10\%$ (high intensity)

Simplification of the computational domain

L. Mezzetti, G. Gustin, D. Rainone – AA. 2009-10

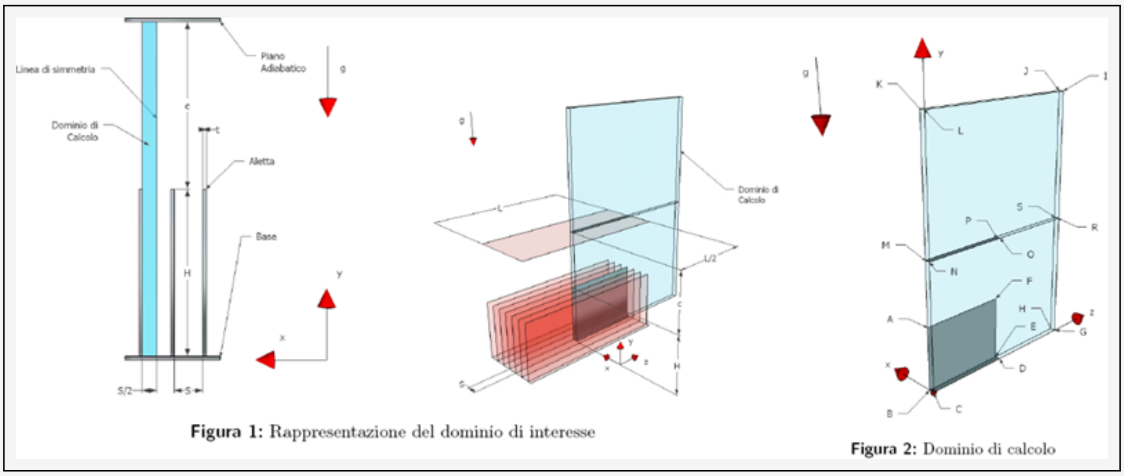


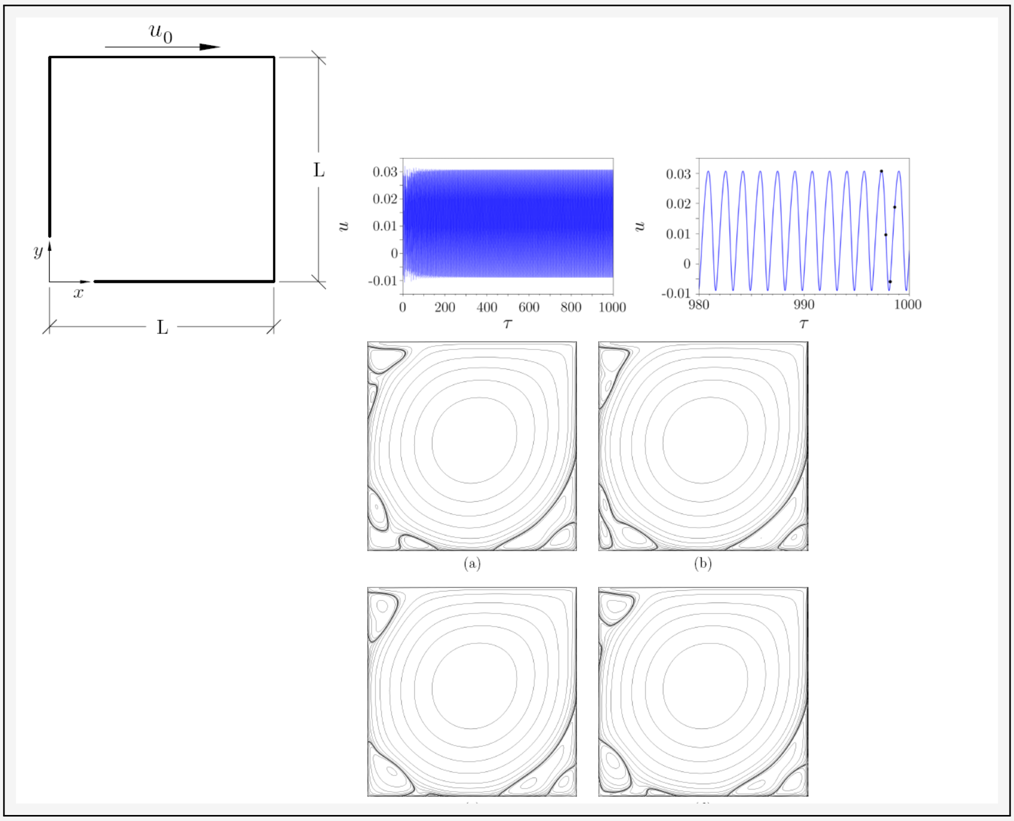
Figura 1: Rappresentazione del dominio di interesse

Figura 2: Dominio di calcolo

Presentation of the courses: Intro to CFD & Comp. meth. for Fluid Dyn. Heat Tr. E. Nobile | Student project | Examples of Student projects

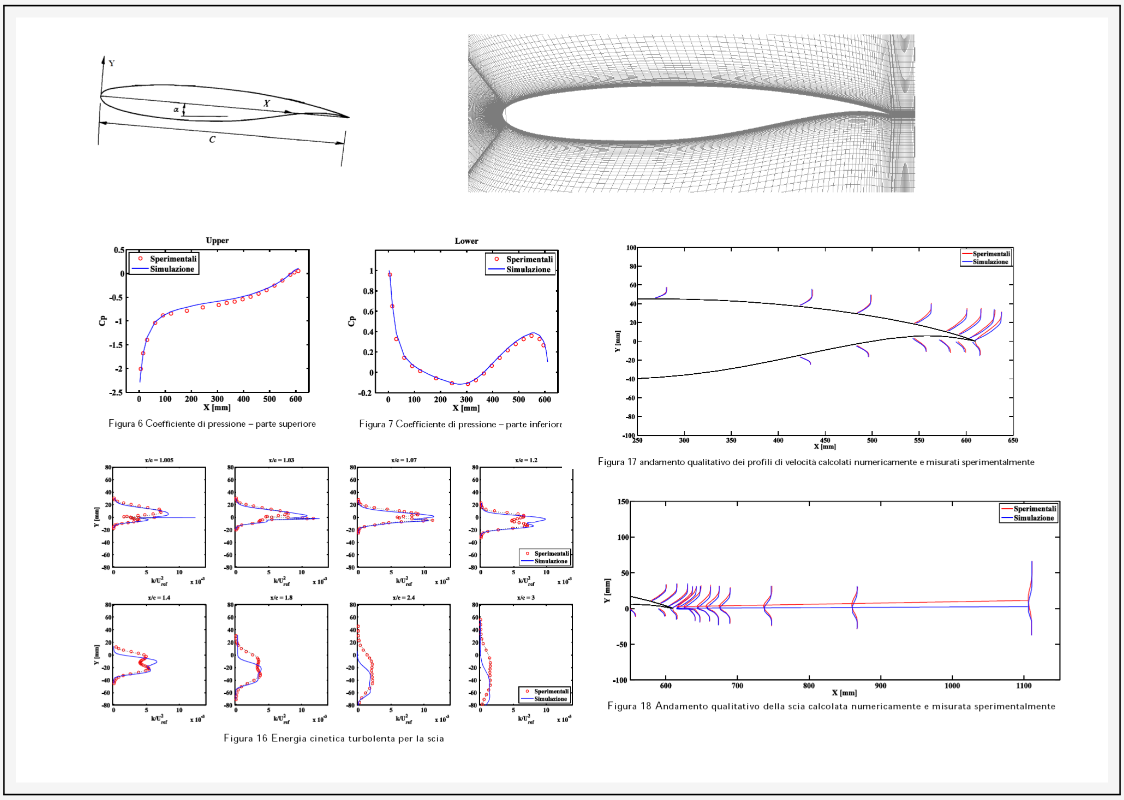
Lid driven cavity

E. Orlandini, R. Zamolo – AA. 2010-11



Flow around a non-symmetrical airfoil

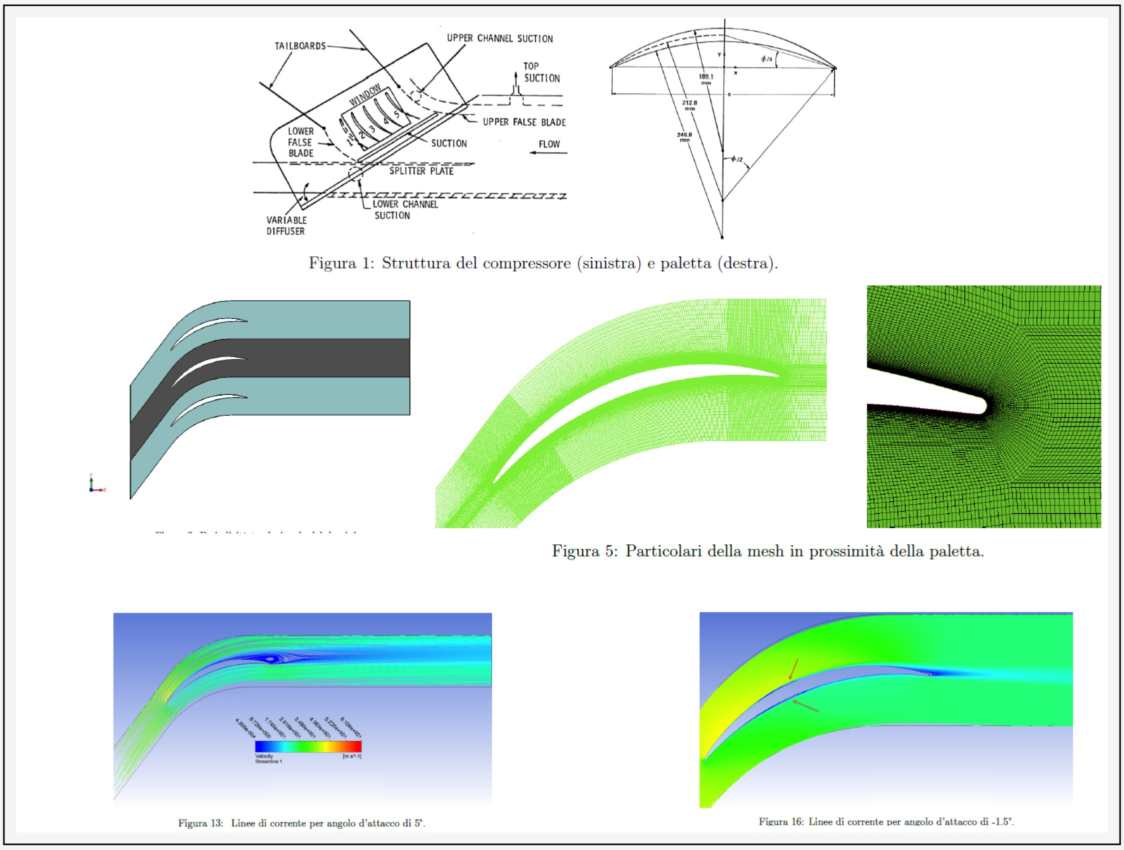
A. Paruta, M. Seriani, M. Turchetto – AA. 2011-12



Presentation of the courses: Intro to CFD & Comp. meth. for Fluid Dyn. Heat Tr. E. Nobile | Student project | Examples of Student projects

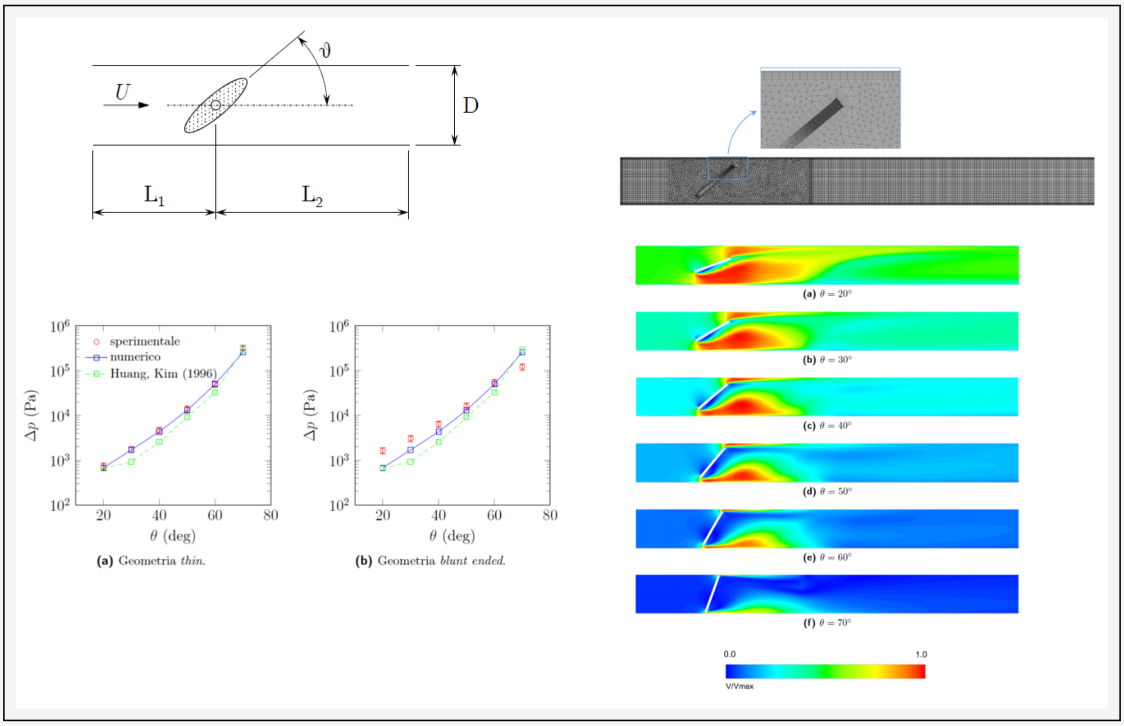
Transitional flow in an axial compressor

D. Battaia, A. Desenibus, P. Marocco – AA. 2011-12



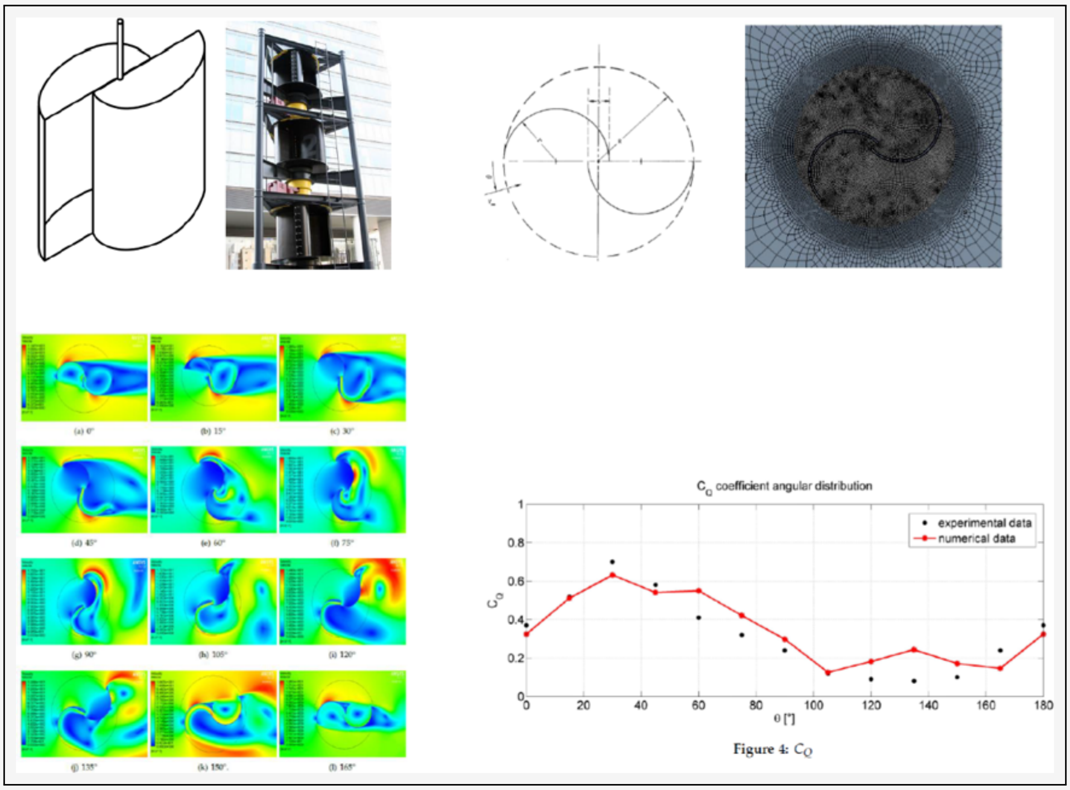
Flow through a butterfly valve

D. Buttignol, M. Macchi, J. Sossi – AA. 2014-15



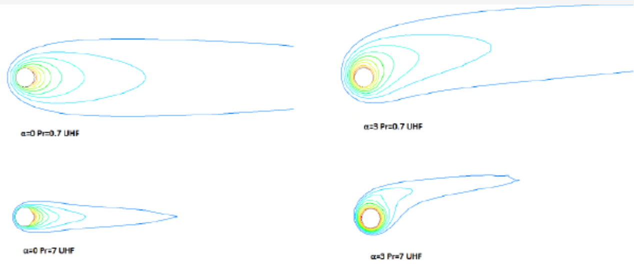
Savonius S wind turbine

L. Battaglia, A. Borzacchiello, G. Scrimali – AA. 2015-16



Heat transfer for a 2D rotating cylinder in laminar regime

V. Daneluzzi, F. Furlan, G. Maggiore – AA. 2016-17



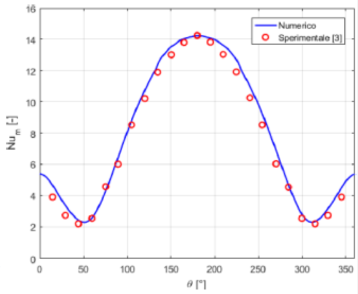


Figura 7: Isoterme per aria (in alto) e acqua (in basso), UHF, con cilindro fermo (a sinistra) e in rotazione (a destra).

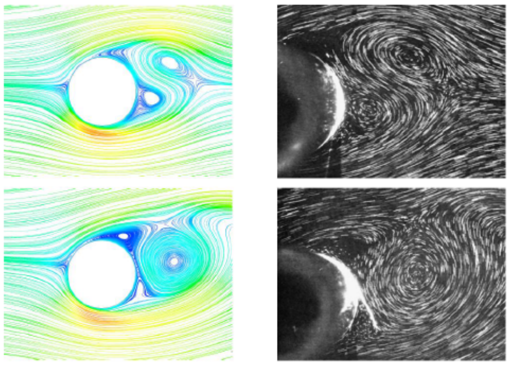
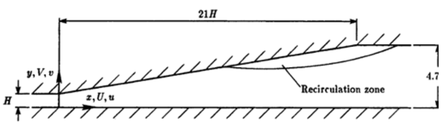
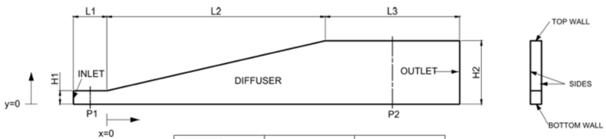


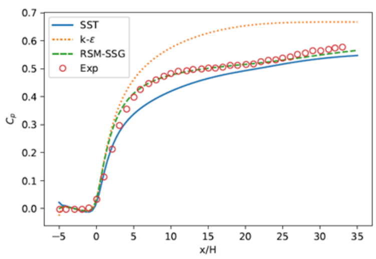
Figura 5: Confronto tra streamlines ottenute per via numerica e sperimentale [5], per $Re = 200, \alpha = 0.5, t^* = 5$ (in alto) e $t^* = 9$ (in basso), con $t^* = 2u_\infty t/D$, CWT.

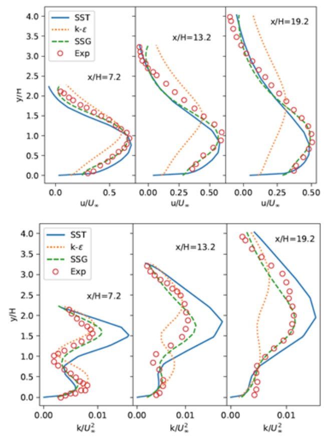
Flow in an asymmetric plane diffuser with different turbulence models

F. De Fazio, M. Ferrari – AA. 2017-18





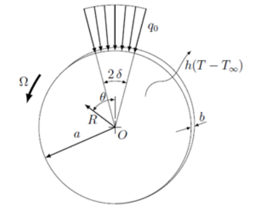


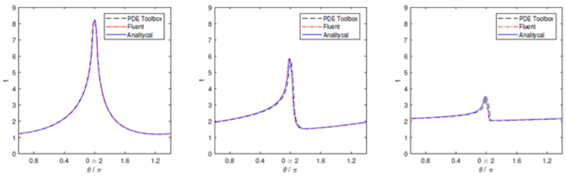


	Turbulence model			Experimental
	SST	k − ε	SSG RSM	
Separation	0.3	-	1.3	1.1
Reattachment	2.9	-	2.4	2.6

Temperature distribution in a thin rotating disk

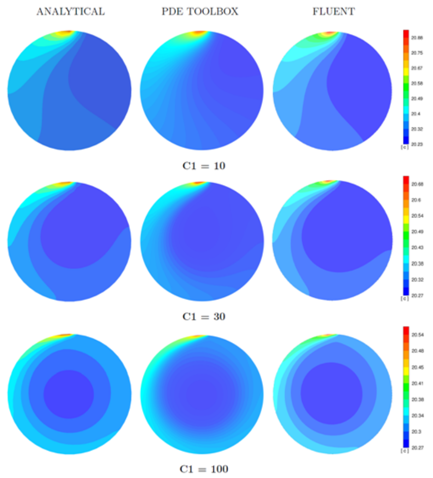
L. Pizzol, F. Carlini – AA. 2018-19





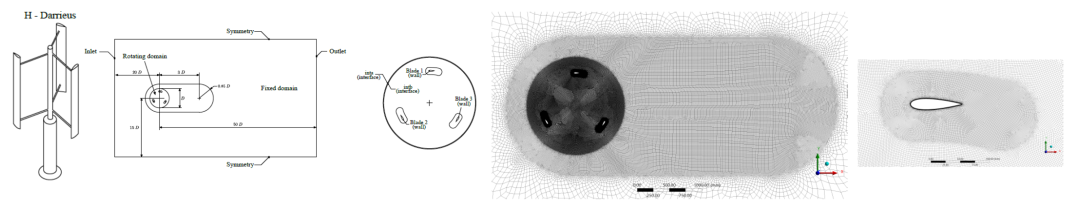
	Analytical Solution	PDE Toolbox	ANSYS Fluent
$C_1 = 1$	8.272	8.234	8.146
$C_1 = 10$	5.876	5.817	5.777
$C_1 = 100$	3.511	3.541	3.486

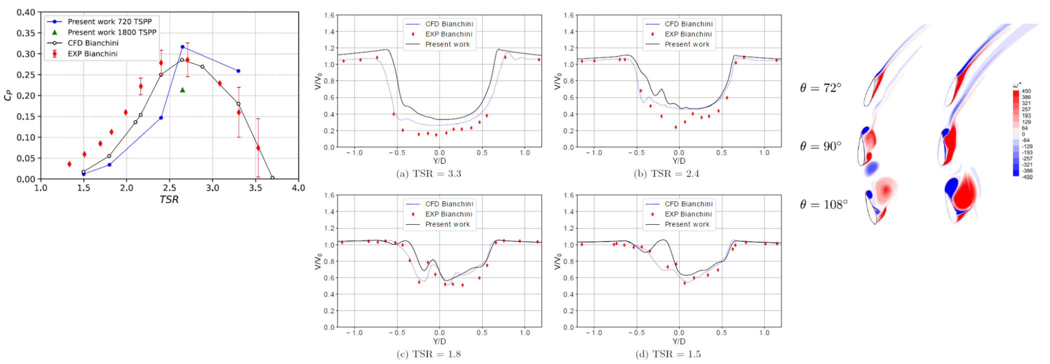
Table 4: Maximum dimensionless temperature



Analysis of a Vertical Axis Wind Turbine

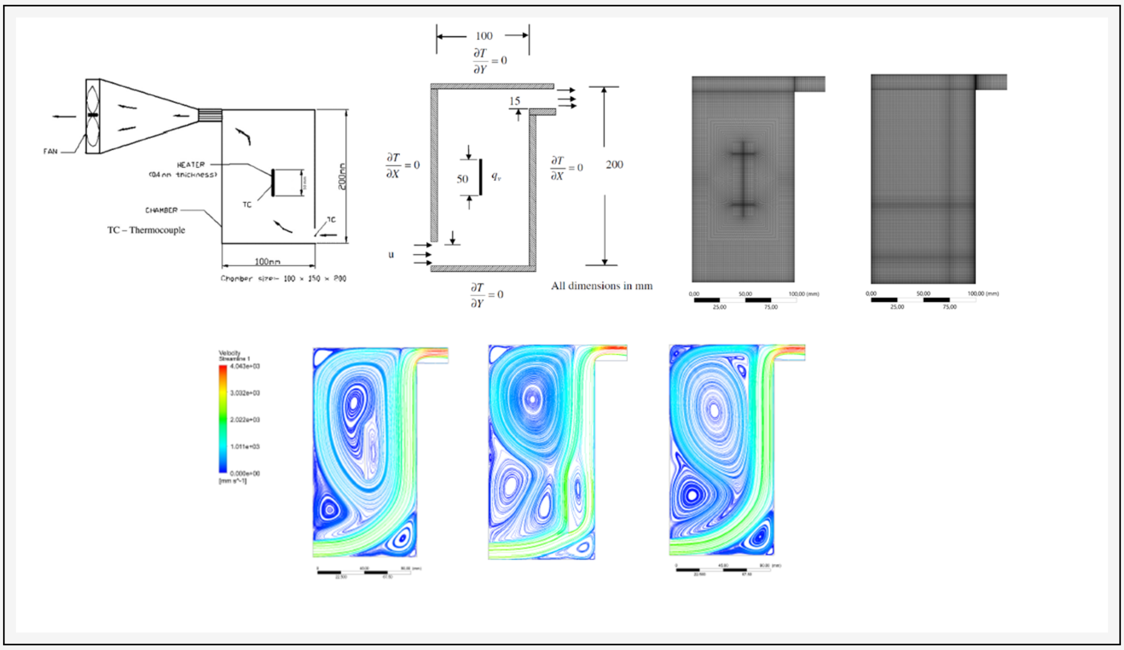
D. Armenante, G. Canever, Riccardo Pavan – AA. 2018-19





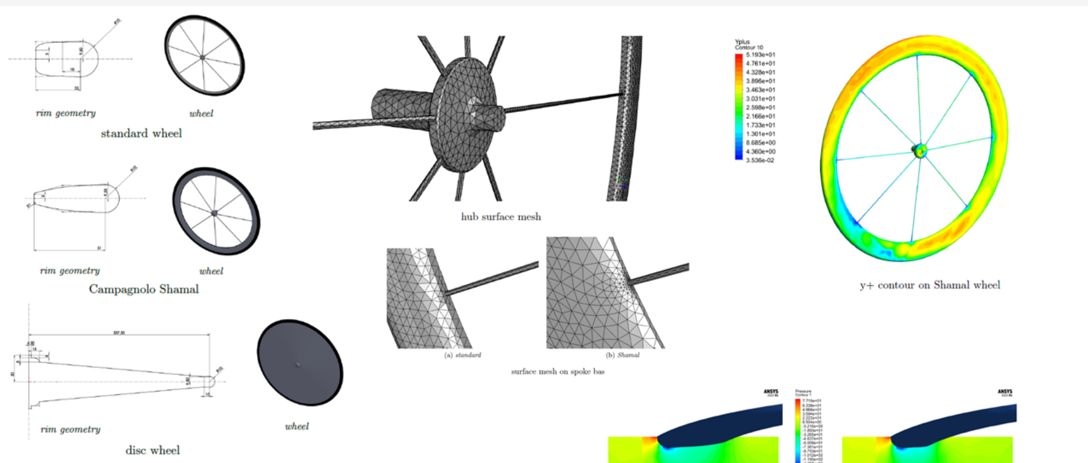
CFD analysis of a ventilated cavity with a heater

J. Pellizon, P. Reggente – AA. 2019-20



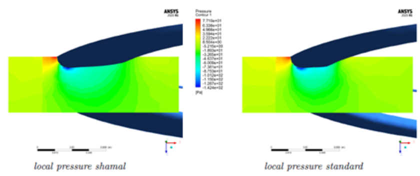
CFD Aerodynamic analysis of different racing cycle wheels

C. Barro Savonuzzi, C. Sanapo – AA 2019-20



wheel	N° elements	Cd	G.S. Tew *et al.* Cd	B. Blocken *et al.* Cd
Standard	509026	0.0298	0.0610	-
Shamal	511166	0.0271	0.0236	0.0259
Disc wheel	506682	0.0243	0.0165	-

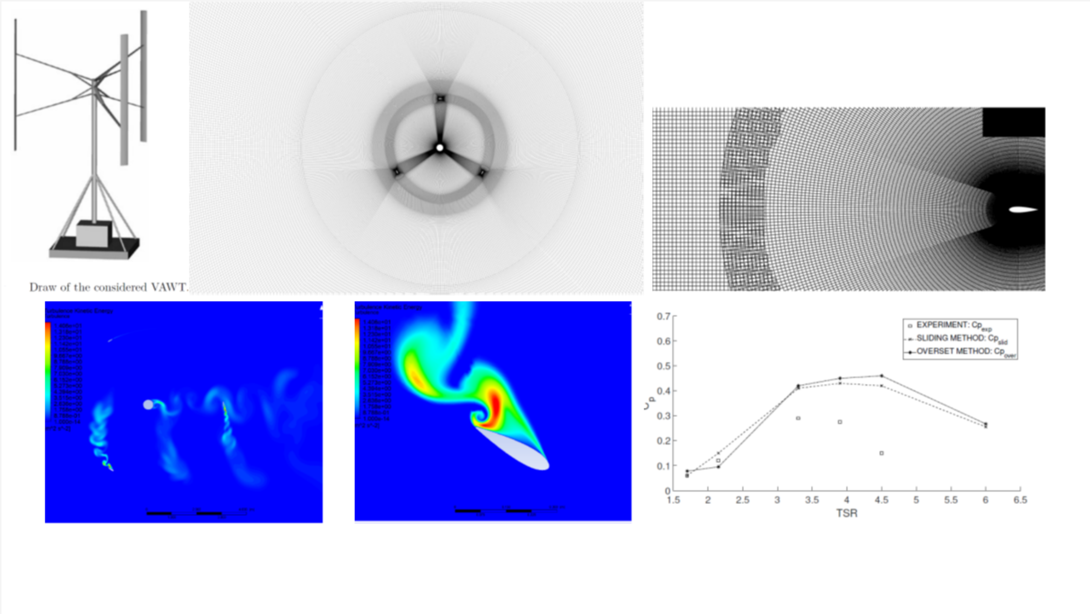
Table : final data



pressure contour plots

SB-VAWT: Comparison between Sliding and Overset Meshing

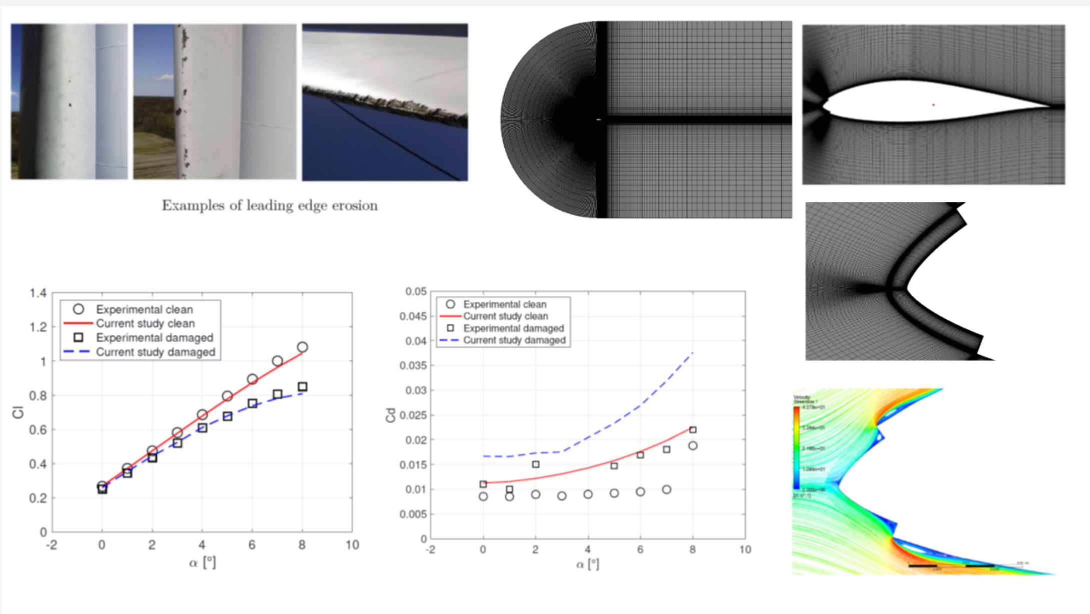
F. Carletti, A. Carroni, D. Valente, G. Valentincic – AA 2021-22



Ansys Fluent: Setting-Up an overset mesh method for a 2D VOWT

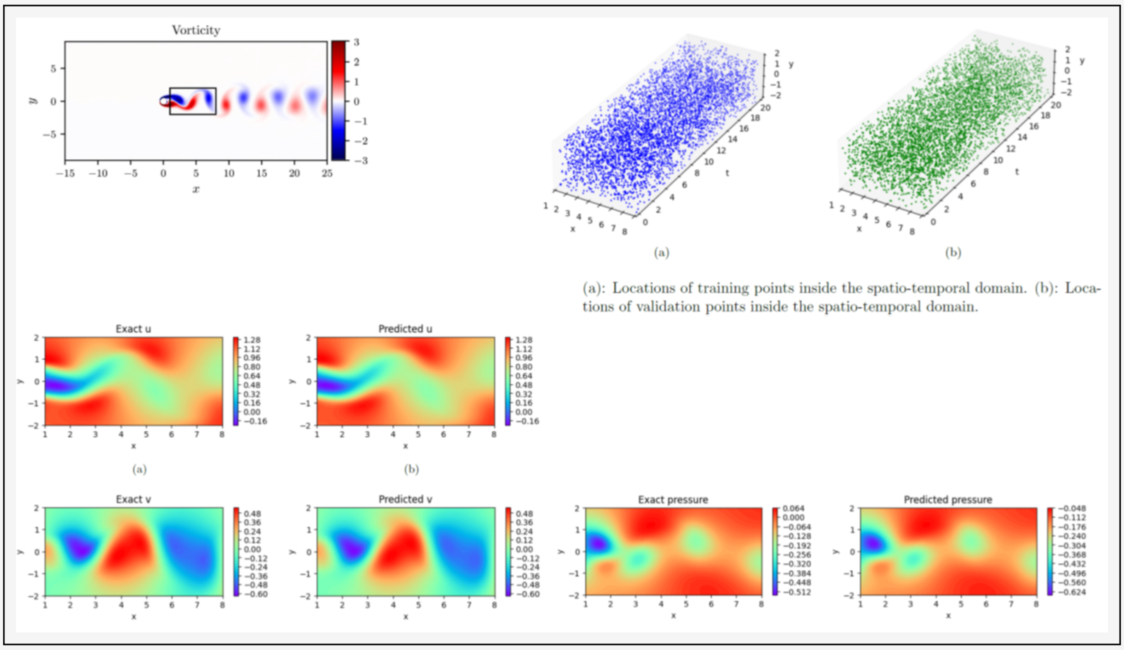
Analysis of Leading Edge Delamination of a DU 96-W-180 Airfoil

M. Accatino, J. Boschian, S. Cettolo – AA 2022-23



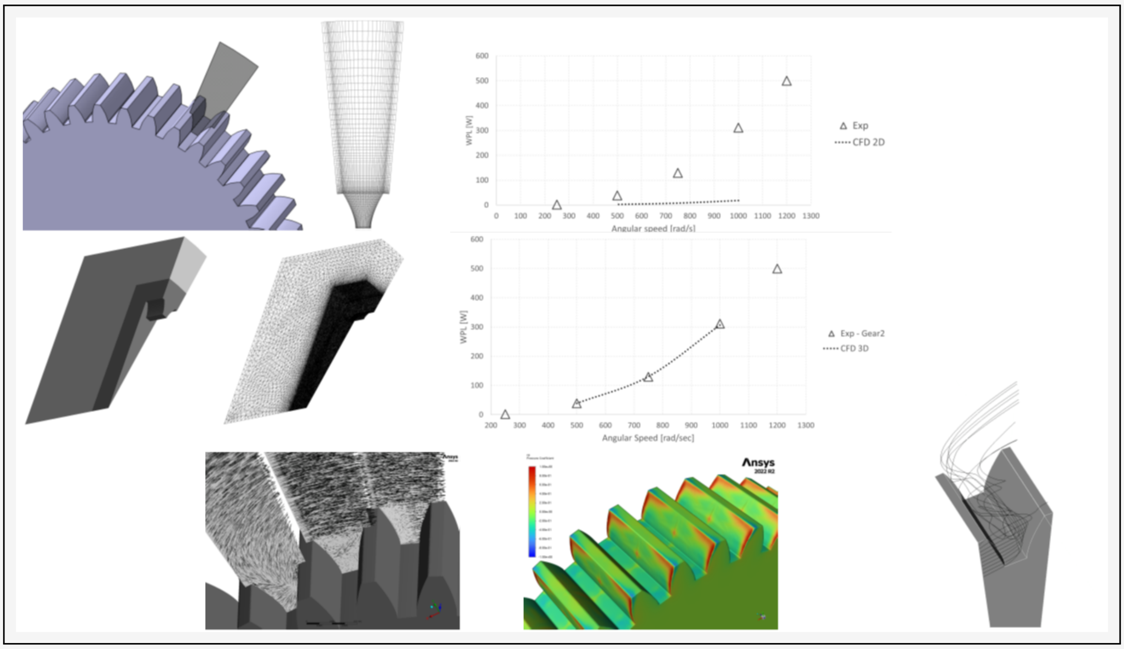
Physics-informed neural networks for solving Navier-Stokes equations: incompressible flow past a circular cylinder

C. Cicala, S. Sancin, A. E. Amodio – AA 2022-23



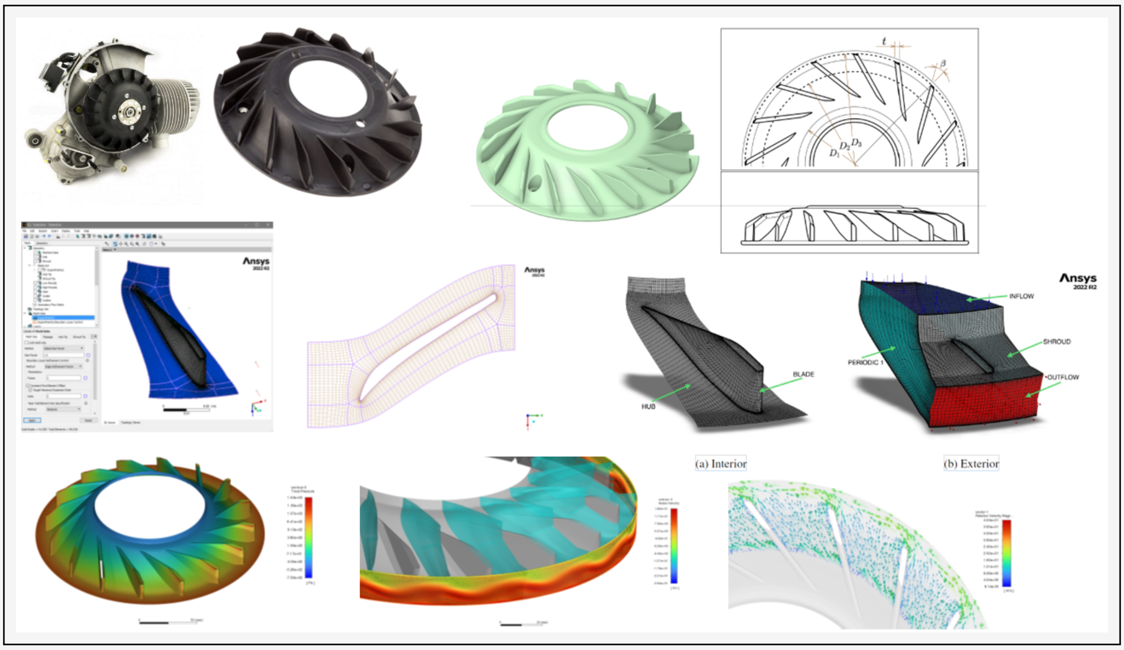
CFD Analysis of gear windage losses

C. La Guardia, S. Puppis – AA 2022-23



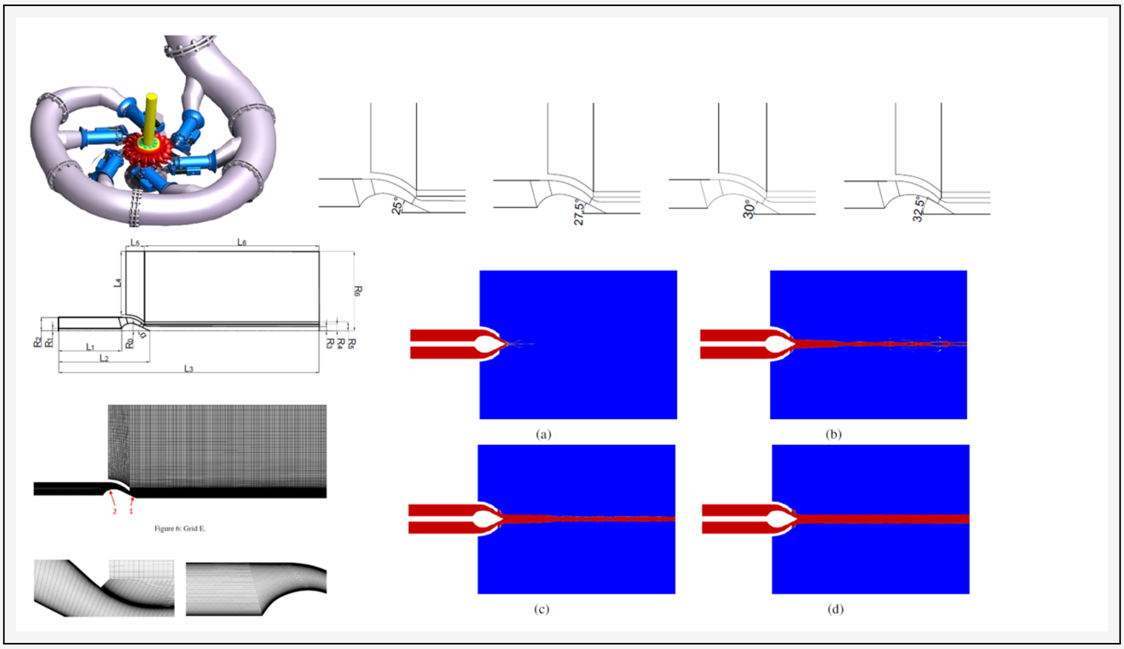
Reverse Engineering and CFD Investigation of Airflow of a Radial Impeller using the SRF Methodology

N. Lorenzon, A. Costa, P. Dariol – AA 2022-23



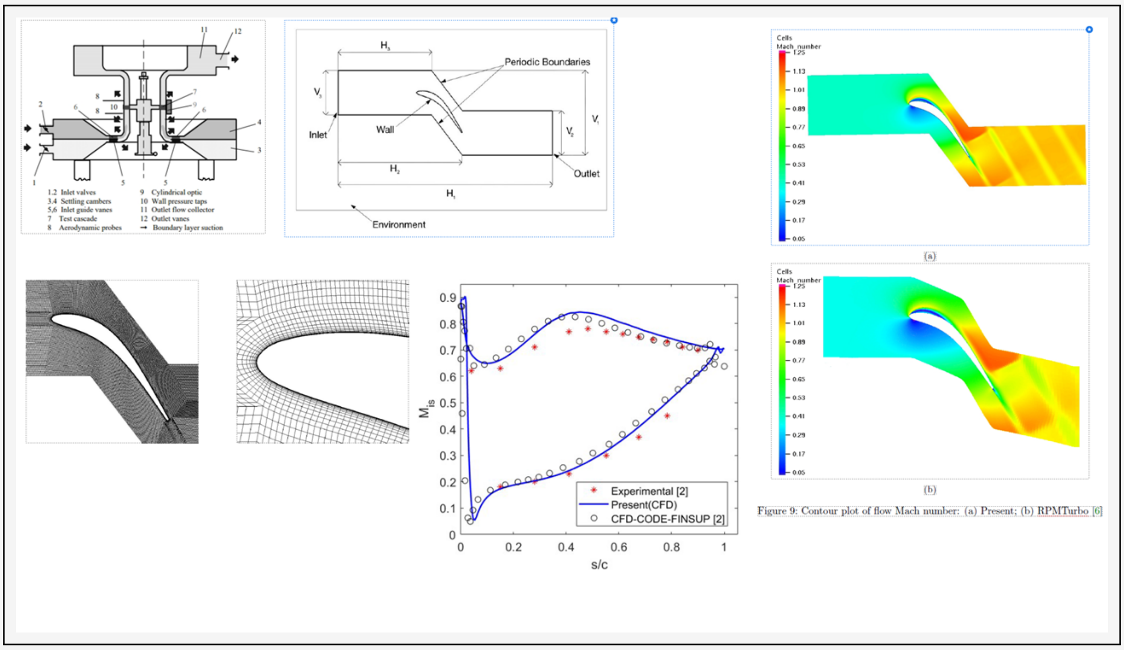
Multiphase Analysis of Velocity Profiles at the Exit of a Pelton Nozzle with Varying Spear Valve Inclination Angles

M. Dibarbora, I. Gregori, N. Prassel – AA 2022-23



CFD Validation of the Steady Case of a Turbine Blade in the 11th Configuration

R. Capizzi, N. Colussi, L. Favero – AA 2023-24



Homework: characteristics

- The Homework will be assigned, from time to time, in written form during class;
- They require the use of some of the methodologies seen in class to solve the proposed problems;
- The student will have to independently develop some simple calculation programs (scripts), using a language of his/her choice:
  - MATLAB suggested (Campus license available);
  - Alternatively Python or Julia;
  - Not recommended: FORTRAN, C/C++ and compiled languages ??in general.
- Unlike the Student project, exercises must be done individually;
- A short (max 8-10 pages) written report, reporting *only the results in graphic and/or tabular form* and any significant observations, must be sent (in PDF) to the professor at least one week before the oral exam:
  - From the 2011/12 academic year it is no longer necessary to bring the paper copy to the oral exam;
  - The teacher has the right to request, before and/or during the oral exam, to examine the programs developed.

Homework: purpose

- Verify the ability to *apply* the notions seen in class through the implementation of single algorithms;
- Increase the capabilities of non-trivial use of commercial and open source CFD software (user subroutines, scripting):
  - Implementation of ad-hoc models, verticalization, automation, etc.
- Become familiar with a programming language (scripting):
  - Know and know how to apply the fundamentals of programming (coding);
  - Multiphysics and/or co-simulation problems, for which it is necessary to develop procedures, via a scripting language, to interface the various programs;
  - Parametric analysis and/or optimization problems;
  - Post-processing and/or specific visualizations.

Some examples from the 2009/10 academic year

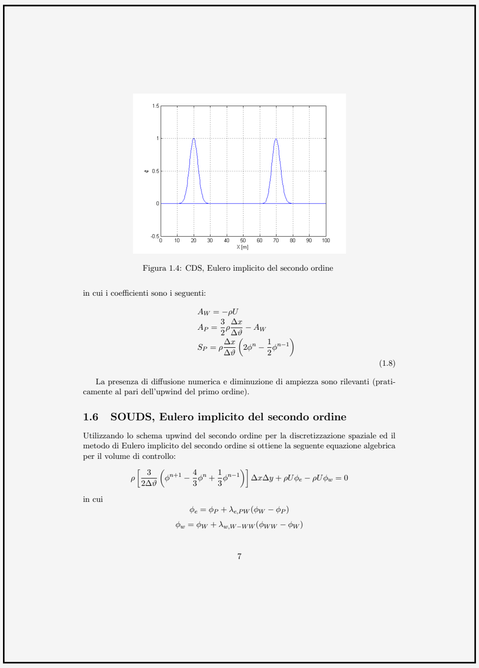


Figura 1.4: CDS, Eulero implicito del secondo ordine

in cui i coefficienti sono i seguenti:

$$A_W = -\rho U$$
$$A_P = \frac{3}{2}\rho\frac{\Delta x}{\Delta\vartheta} - A_W$$
$$S_P = \rho\frac{\Delta x}{\Delta\vartheta}\left(2\phi^n - \frac{1}{2}\phi^{n-1}\right) \quad (1.8)$$

La presenza di diffusione numerica e diminuzione di ampiezza sono rilevanti (praticamente al pari dell'upwind del primo ordine).

1.6 SOUDS, Eulero implicito del secondo ordine

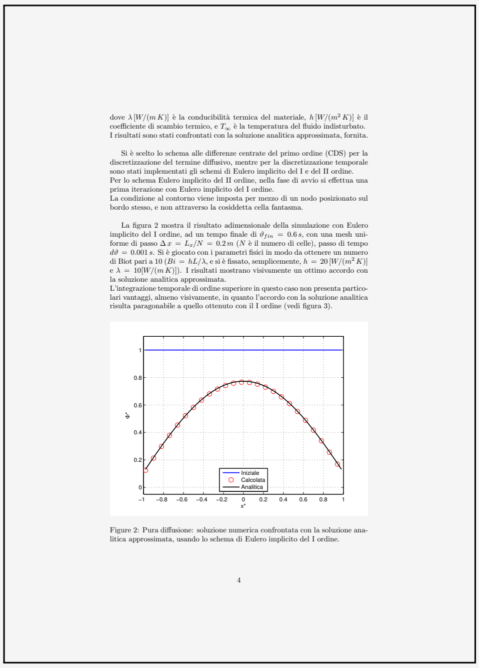
Utilizzando lo schema upwind del secondo ordine per la discretizzazione spaziale ed il metodo di Eulero implicito del secondo ordine si ottiene la seguente equazione algebrica per il volume di controllo:

$$\rho\left[\frac{3}{2\Delta\vartheta}\left(\phi^{n+1} - \frac{4}{3}\phi^n + \frac{1}{3}\phi^{n-1}\right)\right]\Delta x\Delta y + \rho U\phi_e - \rho U\phi_w = 0$$

in cui

$$\phi_e = \phi_P + \lambda_{e,PW}(\phi_W - \phi_P)$$
$$\phi_w = \phi_W + \lambda_{w,W-WW}(\phi_{WW} - \phi_W)$$

7


dove $\lambda\,[W/(m\,K)]$ è la conducibilità termica del materiale, $h\,[W/(m^2\,K)]$ è il coefficiente di scambio termico, e T_∞ è la temperatura del fluido indisturbato. I risultati sono stati confrontati con la soluzione analitica approssimata, fornita.

Si è scelto lo schema alle differenze centrate del primo ordine (CDS) per la discretizzazione del termine diffusivo, mentre per la discretizzazione temporale sono stati implementati gli schemi di Eulero implicito del I e del II ordine.
Per lo schema Eulero implicito del II ordine, nella fase di avvio si effettua una prima iterazione con Eulero implicito del I ordine.
La condizione al contorno viene imposta per mezzo di un nodo posizionato sul bordo stesso, e non attraverso la cosiddetta cella fantasma.

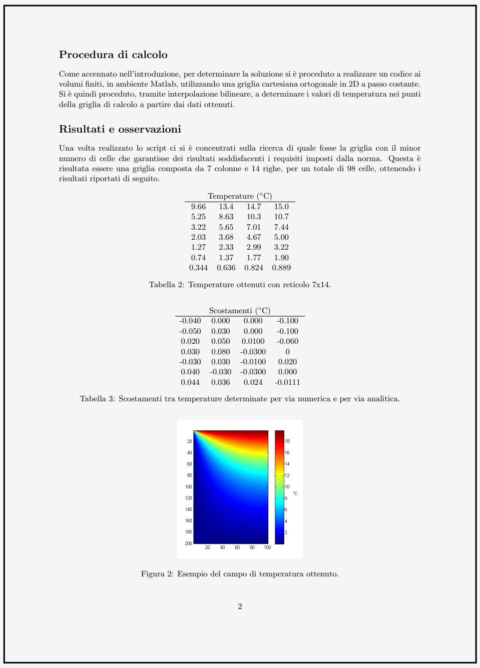
La figura 2 mostra il risultato adimensionale della simulazione con Eulero implicito del I ordine, ad un tempo finale di $\vartheta_{fin} = 0.6\,s$, con una mesh uniforme di passo $\Delta x = L_x/N = 0.2\,m$ (N è il numero di celle), passo di tempo $d\vartheta = 0.001\,s$. Si è giocato con i parametri fisici in modo da ottenere un numero di Biot pari a 10 ($Bi = hL/\lambda$, e si è fissato, semplicemente, $h = 20\,[W/(m^2\,K)]$ e $\lambda = 10[W/(m\,K)]$). I risultati mostrano visivamente un ottimo accordo con la soluzione analitica approssimata.
L'integrazione temporale di ordine superiore in questo caso non presenta particolari vantaggi, almeno visivamente, in quanto l'accordo con la soluzione analitica risulta paragonabile a quello ottenuto con il I ordine (vedi figura 3).

Figure 2: Pura diffusione: soluzione numerica confrontata con la soluzione analitica approssimata, usando lo schema di Eulero implicito del I ordine.

4

Homework AA 2009/10: Purely advective transport of a 1D Gaussian wave and 1D unsteady conduction in a flat plate.

Some examples from the 2010/11 academic year



Procedura di calcolo

Come accennato nell'introduzione, per determinare la soluzione si è proceduto a realizzare un codice ai volumi finiti, in ambiente Matlab, utilizzando una griglia cartesiana ortogonale in 2D a passo costante. Si è quindi proceduto, tramite interpolazione bilineare, a determinare i valori di temperatura nei punti della griglia di calcolo a partire dai dati ottenuti.

Risultati e osservazioni

Una volta realizzato lo script ci si è concentrati sulla ricerca di quale fosse la griglia con il minor numero di celle che garantisse dei risultati soddisfacenti i requisiti imposti dalla norma. Questa è risultata essere una griglia composta da 7 colonne e 14 righe, per un totale di 98 celle, ottenendo i risultati riportati di seguito.

Temperature (°C)			
9.66	13.4	14.7	15.0
5.25	8.63	10.3	10.7
3.22	5.65	7.01	7.44
2.03	3.68	4.67	5.00
1.27	2.33	2.99	3.22
0.74	1.37	1.77	1.90
0.344	0.636	0.824	0.889

Tabella 2: Temperature ottenuti con reticolo 7x14.

Scostamenti (°C)			
-0.040	0.000	0.000	-0.100
-0.050	0.030	0.000	-0.100
0.020	0.050	0.0100	-0.060
0.030	0.080	-0.0300	0
-0.030	0.030	-0.0100	0.020
0.040	-0.030	-0.0300	0.000
0.044	0.036	0.024	-0.0111

Tabella 3: Scostamenti tra temperature determinate per via numerica e per via analitica.

Figura 2: Esempio del campo di temperatura ottenuto.

2

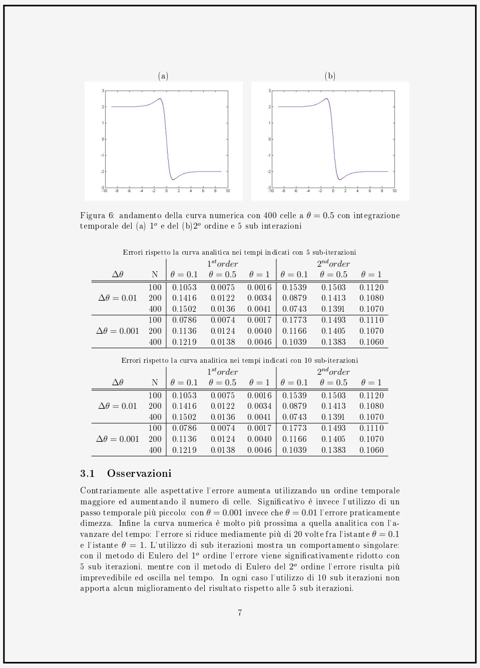


Figura 6: andamento della curva numerica con 400 celle a $\theta = 0.5$ con integrazione temporale del (a) 1° e del (b) 2° ordine e 5 sub interazioni

Errori rispetto la curva analitica nei tempi indicati con 5 sub-iterazioni

$\Delta\theta$	N	$1^{st}order$ $\theta=0.1$	$\theta=0.5$	$\theta=1$	$2^{nd}order$ $\theta=0.1$	$\theta=0.5$	$\theta=1$
	100	0.1053	0.0075	0.0016	0.1539	0.1503	0.1120
$\Delta\theta=0.01$	200	0.1416	0.0122	0.0034	0.0879	0.1413	0.1080
	400	0.1502	0.0136	0.0041	0.0743	0.1391	0.1070
	100	0.0786	0.0074	0.0017	0.1773	0.1493	0.1110
$\Delta\theta=0.001$	200	0.1136	0.0124	0.0040	0.1166	0.1405	0.1070
	400	0.1219	0.0138	0.0046	0.1039	0.1383	0.1060

Errori rispetto la curva analitica nei tempi indicati con 10 sub-iterazioni

$\Delta\theta$	N	$1^{st}order$ $\theta=0.1$	$\theta=0.5$	$\theta=1$	$2^{nd}order$ $\theta=0.1$	$\theta=0.5$	$\theta=1$
	100	0.1053	0.0075	0.0016	0.1539	0.1503	0.1120
$\Delta\theta=0.01$	200	0.1416	0.0122	0.0034	0.0879	0.1413	0.1080
	400	0.1502	0.0136	0.0041	0.0743	0.1391	0.1070
	100	0.0786	0.0074	0.0017	0.1773	0.1493	0.1110
$\Delta\theta=0.001$	200	0.1136	0.0124	0.0040	0.1166	0.1405	0.1070
	400	0.1219	0.0138	0.0046	0.1039	0.1383	0.1060

3.1 Osservazioni

Contrariamente alle aspettative l'errore aumenta utilizzando un ordine temporale maggiore ed aumentando il numero di celle. Significativo è invece l'utilizzo di un passo temporale più piccolo: con $\theta = 0.001$ invece che $\theta = 0.01$ l'errore praticamente dimezza. Infine la curva numerica è molto più prossima a quella analitica con l'avanzare del tempo: l'errore si riduce mediamente più di 20 volte fra l'istante $\theta = 0.1$ e l'istante $\theta = 1$. L'utilizzo di sub iterazioni mostra un comportamento singolare: con il metodo di Eulero del 1° ordine l'errore viene significativamente ridotto con 5 sub iterazioni, mentre con il metodo di Eulero del 2° ordine l'errore risulta più imprevedibile ed oscilla nel tempo. In ogni caso l'utilizzo di 10 sub iterazioni non apporta alcun miglioramento del risultato rispetto alle 5 sub iterazioni.

7

Homework AA 2010/11: *case 1* of the UNI EN ISO 10211 Standard and the 1D Burgers equation.

Some examples from the 2011/12 academic year

Homework AA 2011-2012 – Marco Turchetto

Homework 4: griglia non strutturata

È stata sviluppata in MATLAB una procedura di calcolo che legge entrambi i file tramite il comando importdata, calcola l'area dei singoli elementi e l'area totale. L'area dei singoli elementi è calcolata tramite la regola di Sarrus inserendo in una matrice le coordinate dei vertici dell'elemento triangolare (da consegna). In Tabella 3 sono riportate, solamente per i primi dieci elementi per motivi di spazio, le aree calcolate. L'area totale, ottenuta come somma delle singole aree, è di 0.5 m² (nel caso si interpretino le coordinate riportate nei file in metri).

Le coordinate dei centroidi sono calcolate come media aritmetica delle coordinate dei vertici; nella Tabella 3 sono riportate le coordinate dei centroidi dei primi dieci elementi.

Tabella 3 Area e coordinate dei centroidi per i primi dieci elementi

Area	X	Y	N. elemento	Area	X	Y	N. elemento
0.003160	0.02107	0.05550	1	0.003727	0.02485	0.14913	6
0.003325	0.05440	0.02217	2	0.004262	0.14719	0.02841	7
0.002745	0.10591	0.10463	3	0.004005	0.12578	0.13971	8
0.002950	0.10159	0.05058	4	0.003862	0.05199	0.19784	9
0.003062	0.09310	0.09971	5	0.004147	0.10574	0.19334	10

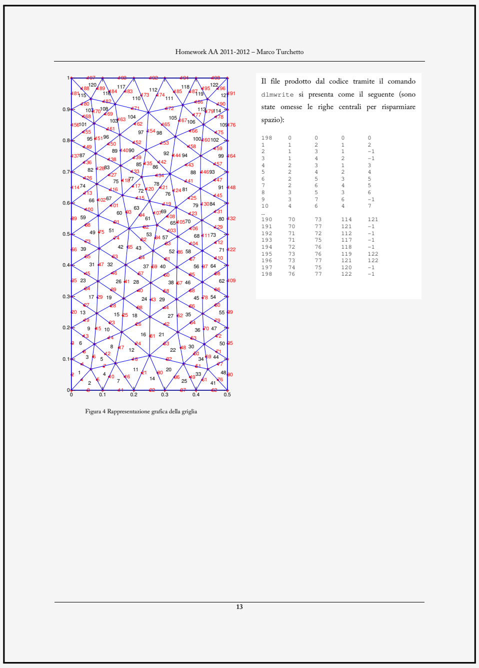
Gli edge della griglia vengono individuati da due nodi di un singolo elemento; dopo aver ordinato (con ciclo for) le coordinate degli edge si utilizza la funzione unique(edge,'rows') per eliminare gli edge considerati più di una volta. Nella tabella seguente sono riportate le coordinate dei centroidi dei primi dieci edge con la relativa numerazione:

Tabella 4 Centroidi dei primi dieci elementi e relativa numerazione

X primo punto	Y primo punto	X secondo punto	Y secondo punto	Numero edge
0.00000	0.00000	0.06321	0.06651	1
0.00000	0.00000	0.00000	0.10000	2
0.00000	0.00000	0.10000	0.00000	3
0.06321	0.06651	0.00000	0.10000	4
0.06321	0.06651	0.10000	0.00000	5
0.06321	0.06651	0.07454	0.14738	6
0.06321	0.06651	0.14156	0.08524	7
0.00000	0.10000	0.07454	0.14738	8
0.00000	0.10000	0.00000	0.20000	9
0.10000	0.00000	0.14156	0.08524	10

Nell'immagine sottostante sono riportati il numero dell'elemento, i nodi e gli edge:

12



Homework AA 2011-2012 – Marco Turchetto

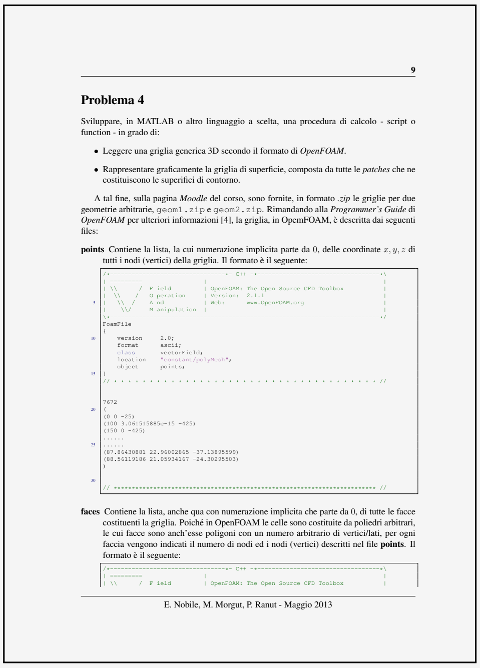
Figura 4 Rappresentazione grafica della griglia

Il file prodotto dal codice tramite il comando dlmwrite si presenta come il seguente (sono state omesse le righe centrali per risparmiare spazio):

13

Homework AA 2011/12: management of unstructured grids.

Some examples from the 2012/13 academic year



9

Problema 4

Sviluppare, in MATLAB o altro linguaggio a scelta, una procedura di calcolo - script o function - in grado di:

- Leggere una griglia generica 3D secondo il formato di *OpenFOAM*.
- Rappresentare graficamente la griglia di superficie, composta da tutte le *patches* che ne costituiscono le superfici di contorno.

A tal fine, sulla pagina *Moodle* del corso, sono fornite, in formato *.zip* le griglie per due geometrie arbitrarie, `geom1.zip` e `geom2.zip`. Rimandando alla *Programmer's Guide* di *OpenFOAM* per ulteriori informazioni [4], la griglia, in OpemFOAM, è descritta dai seguenti files:

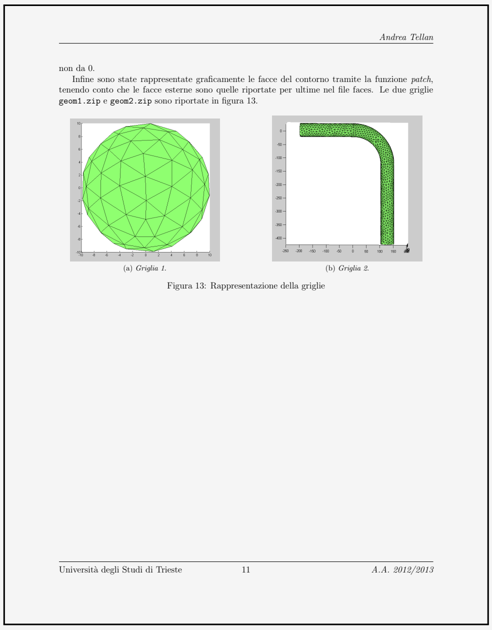
points Contiene la lista, la cui numerazione implicita parte da 0, delle coordinate x, y, z di tutti i nodi (vertici) della griglia. Il formato è il seguente:

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       vectorField;
    location    "constant/polyMesh";
    object      points;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


7672
(
(0 0 -25)
(100 3.061515885e-15 -425)
(150 0 -425)
......
......
(87.86430881 22.96002865 -37.13895599)
(88.56119186 21.05934167 -24.30295503)
)


// ************************************************************************* //
```

faces Contiene la lista, anche qua con numerazione implicita che parte da 0, di tutte le facce costituenti la griglia. Poiché in OpenFOAM le celle sono costituite da poliedri arbitrari, le cui facce sono anch'esse poligoni con un numero arbitrario di vertici/lati, per ogni faccia vengono indicati il numero di nodi ed i nodi (vertici) descritti nel file **points**. Il formato è il seguente:

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
```

E. Nobile, M. Morgut, P. Ranut - Maggio 2013



Andrea Tellan

non da 0.

Infine sono state rappresentate graficamente le facce del contorno tramite la funzione *patch*, tenendo conto che le facce esterne sono quelle riportate per ultime nel file faces. Le due griglie `geom1.zip` e `geom2.zip` sono riportate in figura 13.

(a) *Griglia 1.* (b) *Griglia 2.*

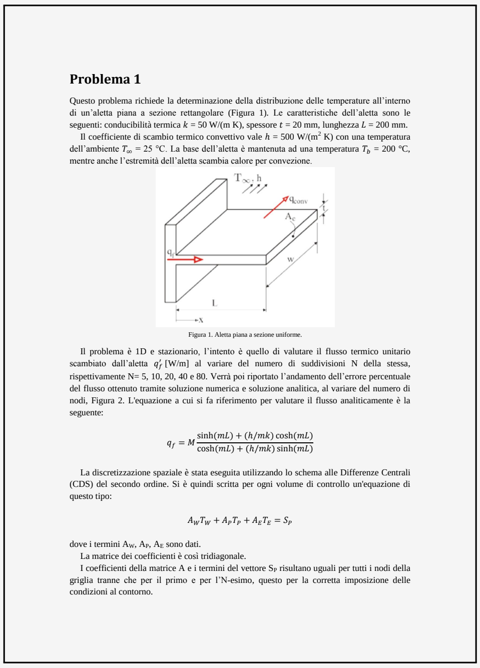
Figura 13: Rappresentazione della griglie

Università degli Studi di Trieste 11 *A.A. 2012/2013*

Homework AA 2012/13: reading and viewing OpenFOAM® grids.

Presentation of the courses: Intro to CFD & Comp. meth. for Fluid Dyn. Heat Tr. E. Nobile | Homework

Some examples from the 2013/14 academic year



Problema 1

Questo problema richiede la determinazione della distribuzione delle temperature all'interno di un'aletta piana a sezione rettangolare (Figura 1). Le caratteristiche dell'aletta sono le seguenti: conducibilità termica k = 50 W/(m K), spessore t = 20 mm, lunghezza L = 200 mm.

Il coefficiente di scambio termico convettivo vale h = 500 W/(m^2 K) con una temperatura dell'ambiente T_∞ = 25 °C. La base dell'aletta è mantenuta ad una temperatura T_b = 200 °C, mentre anche l'estremità dell'aletta scambia calore per convezione.

Figura 1. Aletta piana a sezione uniforme.

Il problema è 1D e stazionario, l'intento è quello di valutare il flusso termico unitario scambiato dall'aletta q'_f [W/m] al variare del numero di suddivisioni N della stessa, rispettivamente N= 5, 10, 20, 40 e 80. Verrà poi riportato l'andamento dell'errore percentuale del flusso ottenuto tramite soluzione numerica e soluzione analitica, al variare del numero di nodi, Figura 2. L'equazione a cui si fa riferimento per valutare il flusso analiticamente è la seguente:

$$q_f = M\frac{\sinh(mL) + (h/mk)\cosh(mL)}{\cosh(mL) + (h/mk)\sinh(mL)}$$

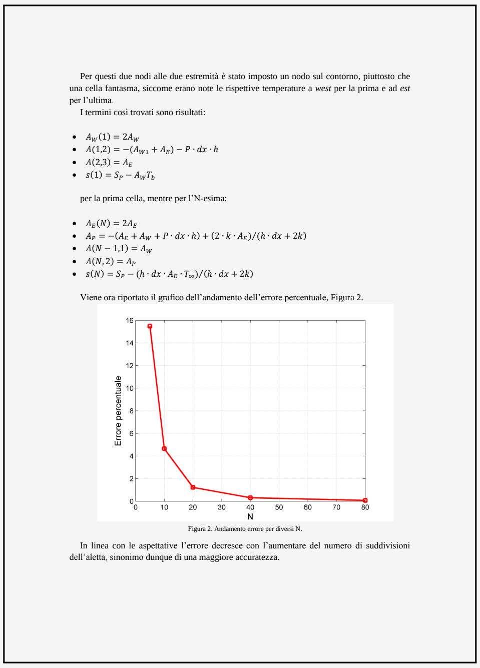
La discretizzazione spaziale è stata eseguita utilizzando lo schema alle Differenze Centrali (CDS) del secondo ordine. Si è quindi scritta per ogni volume di controllo un'equazione di questo tipo:

$$A_W T_W + A_P T_P + A_E T_E = S_P$$

dove i termini A_W, A_P, A_E sono dati.

La matrice dei coefficienti è così tridiagonale.

I coefficienti della matrice A e i termini del vettore S_P risultano uguali per tutti i nodi della griglia tranne che per il primo e per l'N-esimo, questo per la corretta imposizione delle condizioni al contorno.



Per questi due nodi alle due estremità è stato imposto un nodo sul contorno, piuttosto che una cella fantasma, siccome erano note le rispettive temperature a *west* per la prima e ad *est* per l'ultima.

I termini così trovati sono risultati:

- $A_W(1) = 2A_W$
- $A(1,2) = -(A_{W1} + A_E) - P \cdot dx \cdot h$
- $A(2,3) = A_E$
- $s(1) = S_P - A_W T_b$

per la prima cella, mentre per l'N-esima:

- $A_E(N) = 2A_E$
- $A_P = -(A_E + A_W + P \cdot dx \cdot h) + (2 \cdot k \cdot A_E)/(h \cdot dx + 2k)$
- $A(N-1,1) = A_W$
- $A(N,2) = A_P$
- $s(N) = S_P - (h \cdot dx \cdot A_E \cdot T_\infty)/(h \cdot dx + 2k)$

Viene ora riportato il grafico dell'andamento dell'errore percentuale, Figura 2.

Errore percentuale
N

Figura 2. Andamento errore per diversi N.

In linea con le aspettative l'errore decresce con l'aumentare del numero di suddivisioni dell'aletta, sinonimo dunque di una maggiore accuratezza.

Homework AA 2013/14: calculation of heat transfer from a fin.

Some examples from the 2016/17 academic year

Problem 4

Problem description

Consider the simple 2D grid illustrated in figure 8, taken directly from Chapt. 9 in [6]. The coordinates of the mesh nodes (vertices) are shown in the figure and are also given, for convenience, in table 2.

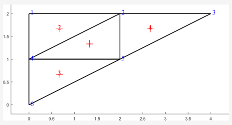


Figura 8: Grid for problem 4.

Node	x	y
1	0.0	2.0
2	2.0	2.0
3	4.0	2.0
4	0.0	1.0
5	2.0	1.0
6	0.0	0.0

Tabella 2: Coordinates of the nodes (vertices) of the grid of figure 8.

The cell definitions are stored in the cell connectivity matrix. This is a matrix of node (vertex) numbers where each row of the matrix contains the connectivity - the nodes - of the corresponding cell. It is given, for this simple case, in table 3.

Cell	Nodes		
1	4	2	5
2	4	1	2
3	6	4	5
4	5	2	3

Tabella 3: Cell connectivity for the grid of figure 8.

The temperatures at the cell centroids are given by

$$T(x,y) = 70\left(x^3 + y^2 + xy + 5\right)$$

Question

Compute the gradient at the centroid of cell *1* using the following two approaches:

1. Cell-based Gauss-Green method (for this problem, since the selected cell faces are mid-way between the centroids of the two attached cells, using weighted interpolation has no effect).
2. Node-based Gauss-Green method.

Compare the results with the exact (analytical) gradient given by

$$\nabla T(x,y) = 70\left(3x^2 + y\right)\mathbf{i} + 70\left(2y + x\right)\mathbf{j}$$

For the calculations it is possible to use either a manual procedure or a computer program. For the latter case, it is useful to develop a code/script (MATLAB, OCTAVE, Scilab or Python recommended) accordingly to the requirements described in problem 4 of Homework 2015-16.

Homework AA 2016/17: gradient calculation for a simple unstructured grid.

Presentation of the courses: Intro to CFD & Comp. meth. for Fluid Dyn. Heat Tr. E. Nobile | Homework

Some examples from the 2017/18 academic year

Problem 3

Problem description

Consider the simple 2D grid constituted by triangular equilateral cells illustrated in figure 1, taken directly from Chapt. 9 in [1]. The coordinates of the mesh nodes (vertices) are provided in the associated file `grid.nod` and are also given, for convenience, in table 1.

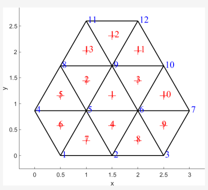


Figure 1: Grid for problem 3.

Node	x	y
1	0.5	0.0
2	1.5	0.0
3	2.5	0.0
4	0.0	0.8660254
5	1.0	0.8660254
6	2.0	0.8660254
7	3.0	0.8660254
8	0.5	1.7320508
9	1.5	1.7320508
10	2.5	1.7320508
11	1.0	2.5980762
12	2.0	2.5980762

Table 1: Coordinates of the nodes (vertices) of the grid of figure 1.

The cell definitions are stored in the cell connectivity matrix. This is a matrix of node (vertex) numbers where each row of the matrix contains the connectivity - the nodes - of the corresponding cell. It is given in the file `grid.ele` and also in table 2.

Cell	Nodes		
1	5	6	9
2	5	9	8
3	6	10	9
4	2	6	5
5	4	5	8
6	1	5	4
7	1	2	5
8	2	3	6
9	3	7	6
10	6	7	10
11	9	10	12
12	9	12	11
13	8	9	11

Table 2: Cell connectivity for the grid of figure 1.

Question

Consider the three following cases of temperature distribution, and corresponding (exact) analytical expression of the gradient:

Linear

$$T(x,y) = 100x + 30y + 5$$
$$\nabla T(x,y) = 100\,\mathbf{i} + 30\,\mathbf{j}$$

Quadratic

$$T(x,y) = 100\left(x^2 + y^2 + 1\right)$$
$$\nabla T(x,y) = 200x\,\mathbf{i} + 200y\,\mathbf{j}$$

Cubic

$$T(x,y) = 100\left(x^3 + y^2 + xy + 5\right)$$
$$\nabla T(x,y) = 100\left(3x^2 + y\right)\mathbf{i} + 100\left(2y + x\right)\mathbf{j}$$

Compute numerically, for the three temperature distributions, the temperature gradient at the centroid of cell *1* using the cell-based Gauss-Green method [1] and compare the results with the exact gradient.

For the calculations develop a code/script (MATLAB, OCTAVE, Scilab or Python recommended) accordingly to the requirements described in problem 4 of Homework 2015-16.

[1] For this problem, since the cell faces are midway between the centroids of the two attached cells, using weighted interpolation has no effect.

Homework AA 2017/18: unstructured grid gradient reconstruction for different temperature distributions.

Some example from the 2018/19 academic year

Problem 3

Problem description

Consider the two simple 2D grids constituted by triangular cells illustrated in figures 1 and 2. The first, *regular* one constituted by equilateral triangles, has been taken directly from Chapt. 9 in [1]. The second, *irregular* one, is a *deformed* version of the former. The coordinates of the mesh nodes (vertices) are provided in the associated files `grid1.nod` and `grid2.nod`, and are also given, for convenience, in tables 1 and 2, respectively.

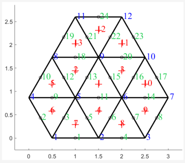


Figure 1: Grid 1 for problem 3.

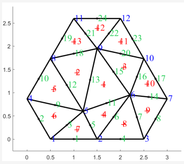


Figure 2: Grid 2 for problem 3.

Node	x	y
1	0.5	0.0
2	1.5	0.0
3	2.5	0.0
4	0.0	0.8660254
5	1.0	0.8660254
6	2.0	0.8660254
7	3.0	0.8660254
8	0.5	1.7320508
9	1.5	1.7320508
10	2.5	1.7320508
11	1.0	2.5980762
12	2.0	2.5980762

Table 1: Coordinates of the nodes (vertices) of the grid of figure 1.

Node	x	y
1	0.5	0.0
2	1.5	0.0
3	2.5	0.0
4	0.0	0.8660254
5	1.2	0.6000000
6	2.2	0.9500000
7	3.0	0.8660254
8	0.5	1.7320508
9	1.5	1.9500000
10	2.5	1.7320508
11	1.0	2.5980762
12	2.0	2.5980762

Table 2: Coordinates of the nodes (vertices) of the grid of figure 2.

The cell definitions are stored in the cell connectivity matrix which, in this problem, is the same for both grids. This is a matrix of node (vertex) numbers where each row of the matrix contains the connectivity - the nodes - of the corresponding cell. It is given in the file `grid.ele` and also in table 3.

Question

Consider the three following cases of temperature distribution, and corresponding (exact) analytical expression of the gradient:

Linear

$$T(x,y) = 100x + 30y + 5$$
$$\nabla T(x,y) = 100\,\mathbf{i} + 30\,\mathbf{j}$$

Homework AA 2018/19: gradient reconstruction for orthogonal and non-orthogonal unstructured grid.

Some example from the 2019/20 academic year

Proposed problem

A plane fin of uniform cross-section, as shown in figure 1, is made with a uniform, isotropic material with a thermal conductivity $k = 50$ W/(m K); it has a thickness t, a length L and its width w is large in comparison with its thickness. In this case the fin is cooled by *both* convection and radiation:

- The convective heat transfer coefficient is h =50 W/(m^2 K) and the surrounding fluid temperature is T_∞.
- The radiation heat transfer can be accounted for assuming that the fin surface is gray-diffuse, with a global emissivity ϵ = 0.7, that the fluid is not participating nor emitting, and that the temperature of the surroundings is T_s.

The base of the fin is maintained at a temperature $T_b = 400$ °C, while the tip of the fin can be assumed perfectly insulated.

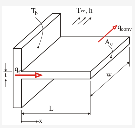


Figure 1: Straight fin with uniform cross-section.

Assuming a *1D* temperature distribution, i.e. $T \approx T(x)$, compute, with the Finite Volume method (FV), the heat flux per unit width of the fin q'_{num} [W/m], using a number N of FVs equal to $N = 10$, 20, 40, and 80.

Consider the three following cases:

1. In this first case the fin has a length $L = 20$ mm and a thickness $t = 1$ mm, the fluid temperature and the surroundings temperature are the same, i.e. $T_\infty \equiv T_s$ = -273.15 °C = 0 K. In this way, it is possible to verify the result by comparison with the simplified analytical expression given in [1], not reported here for brevity, which gives q'_{appr} = 1186.0 [W/m].
2. For the second case, repeat the calculation but assume T_∞ = 25 °C and T_s = 15 °C.
3. For the third case, consider a fin with a different geometry (lower aspect-ratio): $L = 20$ mm and thickness $t = 10$ mm. All other data are the same as in the second case.

TIP # 1

Due to radiation heat transfer the problem is *non-linear*, and therefore it requires an iterative approach with a proper linearization. In this case it can be done by observing that the heat transfer by radiation can be expressed in a form similar to that of convection heat transfer:

$$q'_{rad} = \epsilon\sigma A_f \left(T_f^4 - T_s^4\right)$$
$$= A_f h_r \left(T_f - T_s\right)$$

where T_f is the (local) temperature of the fin, $\sigma = 5.670367 \times 10^{-8}$ W/(m^2 K^4) is the Stefan-Boltzmann constant and

$$h_r = \epsilon\sigma \left(T_f^2 + T_s^2\right)\left(T_f + T_s\right)$$

is the *radiation heat transfer coefficient*. Since h_r *depends* on T_f, an iterative procedure, with a check of convergence, should be applied.
In this way, the total heat transfer can be expressed as

$$q'_{tot} = q'_{conv} + q'_{rad} = A_f \left[h\left(T_f - T_\infty\right) + h_r\left(T_f - T_s\right)\right]$$

Needless to say, proper attention must be paid, in this case, to the unit of measure of temperature.

TIP # 2

In order to verify, at least partially, the correctness of the script, a possible solution is to switch off the radiation heat transfer contribution, and compare the numerical result with the analytical solution [2] valid for longitudinal fins of uniform cross-section with insulated tip:

$$q'_f = \sqrt{2hkt}\,tanh(mL)\left(T_b - T_\infty\right)$$

with

$$m = \sqrt{2h/kt}$$

References

[1] Y. Huang, X.-F. Li, Exact and approximate solutions of convective-radiative fins with temperature-dependent thermal conductivity using integral equation method, *Int. J. Heat Mass Transfer*, **150**, 119303, 2020.

[2] F. P. Incropera, D. P. Dewitt, T. L. Bergman, A. S. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th Ed., Wiley, (2007).

Homework AA 2019/20: heat transfer by convection and radiation from a 1D fin.

Some example from the 2020/21 academic year

Proposed problem

An annular fin of rectangular profile, as shown in figure 1, is made with a uniform, isotropic material with a thermal conductivity $k = 40$ [W/(m K)]; it has a thickness $t = 10$ [mm], an inner radius $r_1 = 50$ [mm] and an outer radius $r_2 = 125$ [mm].

The fin is cooled only by convection, with a convective heat transfer coefficient $h = 40$ [W/(m^2 K)], and the surrounding fluid temperature $T_\infty = 25$ [°C]. The base of the fin is maintained at a temperature $T_b = 110$ [°C], while also the tip of the fin contribute, with the same heat transfer coefficient, to the overall heat flux.

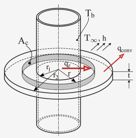


Figure 1: Annular (radial) fin of rectangular profile.

Assuming a *1D* temperature distribution, i.e. $T \approx T(r)$, compute, with the Finite Volume method (FV), the heat flux from the fin q_{num} [W], using a number N of FVs equal to $N = 10$, 20, 40, and 80.
Compare the numerical results with the analytical (exact) solution given in [1]:

$$q_f = 2\pi k r_1 t m \left(T_b - T_\infty\right) \frac{\gamma K_1(mr_1) - I_1(mr_1)}{I_0(mr_1) + \gamma K_0(mr_1)} \quad (1)$$

with

$$\gamma = \frac{(h/mk) I_0(mr_2) + I_1(mr_2)}{K_1(mr_2) - (h/mk) K_0(mr_2)} \quad (2)$$

and

$$m = \sqrt{2h/kt} \quad (3)$$

In (1) and (2) I_0 e I_1 represent, respectively, the modified Bessel function of the first kind of *order 0* and of *order 1* (in MATLAB: function `besseli`), while K_0 e K_1 are the modified

Bessel function of the second kind of *order 0* and of *order 1*, respectively (in MATLAB: function `besselk`).

TIP

Discretize the fin, assuming $T \approx T(r)$, with circular (ring-shaped) Finite Volumes of thickness t and width Δr, noting that, differently from linear fins, the cross-section and the external surface of each FV varies with r, i.e. $A_c = A_c(r)$ and $\Delta A_s = \Delta A_s(r)$.

References

[1] A. D. Kraus, A. Aziz, J. Welty, EXTENDED SURFACE HEAT TRANSFER, J. Wiley & Sons, (2001).

Homework AA 2020/21: heat transfer by convection from an annular fin.

Some example from the 2021/22 academic year

Proposed problem

1. Following *Homework 1 - AA. 2021-22*, first case, consider a cylindrical (pin) fin, as shown in figure 1, which is made with a uniform, isotropic material with a thermal conductivity value of $k = 40$ W/(m K). The fin has a length $L = 40$ mm and a diameter $d = 4$ mm. The fin is cooled only by convection with a convective heat transfer coefficient h =400 W/(m^2 K), and the temperature of the surrounding fluid is $T_\infty = 25$ °C. The temperature of the base of the fin is maintained at a temperature $T_b = 200$ °C, while also the tip of the fin contributes, with the same heat transfer coefficient, to the overall heat flux.

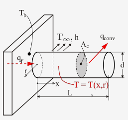


Figure 1: Axisymmetrical cylindrical (pin) fin.

In this case, disregard the usual assumption of 1D temperature distribution (see [1, 2]), i.e.

$$T \approx T(x)$$

and consider a full *2D, axisymmetric* temperature distribution [1, 2]:[1]

$$T = T(x, r)$$

Using the MATLAB *PDE Toolbox*, develop a 2D axisymmetric steady numerical model for the fin and, using an *adequate* number of finite elements, compute the heat flux q_{num2D} [W]. Compare the result with that obtained with the 1D model of *Homework 1*.
What is the $\%$ error using the 1D assumption?
Plot a contour map of the temperature field.

[1] The general heat (conduction) equation for an isotropic material in cylindrical coordinates is

$$\frac{1}{r}\frac{\partial}{\partial r}\left(kr\frac{\partial T}{\partial r}\right) + \frac{1}{r^2}\frac{\partial}{\partial \phi}\left(k\frac{\partial T}{\partial \phi}\right) + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \dot{q}_g = \rho\, c_p \frac{\partial T}{\partial \tau}$$

which, under the assumption of steady, 2D axisymmetric temperature field with no heat generation, reduces to

$$\frac{1}{r}\frac{\partial}{\partial r}\left(kr\frac{\partial T}{\partial r}\right) + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) = 0$$

2. Repeat the same analysis for the second case of *Homework 1*, e.g. fin length $L = 40$ mm and diameter $d = 20$ mm, and again compare the result with that from the 1D model.
What is the $\%$ error using the 1D assumption in this second case? Is it lower or higher? Why?
Plot a contour map of the temperature field.

References

[1] G. Comini, G. Cortella, *Fondamenti di trasmissione del calore*, 4a Ed., S.G.E. Editore, (2013).

[2] F. P. Incropera, D. P. Dewitt, T. L. Bergman, A. S. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th Ed., Wiley, (2007).

Homework AA 2021/22: heat transfer by convection from a 2D (axisymmetrical) cylindrical fin.

Exam

WATCH OUT!

- Students attending only the course *Introduction to Computational Fluid Dynamics* - i.e. students enrolled in the *Engineering for the Energy Transition* master's degree program - are required to submit only a simple student project:
  - Lack of knowledge of some fundamental aspects of CFD suggests selecting moderately simple projects.
  - The topic of the project, in this case, will be suggested by the teacher.
- For them the exam will consists only of an oral discussion of their project.

Exam - *cont.*

- For students attending both courses *Introduction to Computational Fluid Dynamics* and *Computational methods for Fluid Dynamics and Heat Transfer* - i.e. students enrolled in the *Mechanical Engineering* master's degree program - the exam consists of an oral interview, with discussion of the Student project, possible clarifications on the Homework completed and questions on course topics;
- The questions are related to ALL the program carried out:

  It is recommended not to *skip* parts covered in class that are asked less frequently in the exam;
- The objective of the oral exam is to verify the student's knowledge of the fundamental aspects of CFD, and together with the Student project and the Homework to be carried out it should constitute a balanced and homogeneous verification of the knowledge of the various aspects - theoretical and applied - of the discipline.

Examination - instructions

SUMMARY

- Deadlines:
  1. Homeworks: at least 7 days before the exam. Single PDF document with all HW, no attachments (MATLAB scripts etc).
  2. Student project: at least 10 days before the exam. Single PDF document only, no attachments (WB project etc.)
  3. Exam registration (ESSE3): at least 3 days before the exam.
- For the Student Project:
  - Always include project colleagues in related communications/emails.
  - Bring only one paper copy the first time any of the students in the team takes the exam.
  - Double-sided printing, simple stapling, no binding.
- For the Homework:
  - Failure to adhere to the *Rules and suggestions for writing the Final Paper (Student Project) and Homework*.
  - Avoid copying text and images of the assigned HW in the paper (rather make a freehand sketch).

Difficulties

1. Theory
   - Incomplete and/or missing preparation on some theoretical aspects covered in class (i.e. FEM, FD, turbulence modeling, V&V etc.).
2. Student project
   - Sometimes little attention is paid to the quality of the results and the reliability of the reference source of the problem to be solved.
   - Little attention to fundamental aspects (characteristics of the problem, relevant phenomena), and excessive emphasis on the use of the IT tool: Compute less, think more!.
   - Inadequate attention to physical aspects, and insufficient bibliographic research in the assigned project:
     - Difficulty in recognizing the most important physical aspects before starting modeling activities;
     - Difficulty in identifying possible simplification, i.e. 3D $\rightarrow$ 2D, symmetry planes etc.
   - Poor problem-solving aptitude:
     - lack of autonomy and *courage* in facing problems - of various kinds - that may arise during the project.
   - Failure to use the *Rules and suggestions for writing the Student Project report and Homework*, available on Moodle.
3. Homework
   - Difficulty and limited ability to translate what was covered in class into *code* (e.g. MATLAB).
   - Failure to use the *Rules and suggestions for writing the Student Project report and Homework*, available on Moodle.

News AA 2024/25

- To encourage students to take the exam from the first available session:
  - *Homework* proposed as soon as possible, to avoid overloading of work in the final stages of the course.
  - Selection and examination of the *Student project* during the exercises and, where possible, its (partial) implementation in the classroom.
  - TENTATIVE Short presentation (1-2 slides) of the selected student projects at the end of the course.
- Plagiarism check with Turnitin (`https://www.turnitin.com/`).
- From the 2017-18 academic year: the student project must be written in English.
- NEW: update of part of the teaching material (Finite Volume Method for unstructured grids, tutorial(s), homework, transition to ANSYS Rel. 2024 R1 or Rel. 2025 R1).
- Interactive implementation in class - *Live Script* - of some simple exercises with MATLAB and MATLAB/PDEToolbox.
- NEW: Revised *V&V* (Verification and Validation)

News AA 2024/25 - *cont.*

Since the 2018/19 academic year no more introductory lessons on MATLAB have been held but, possibly, only some specific lessons, aimed at solving the problems seen in class. Indeed:

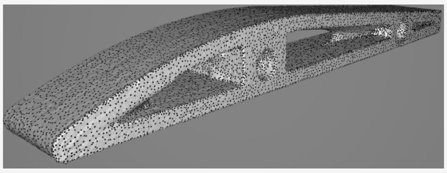
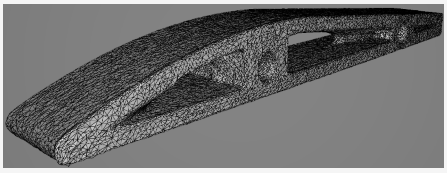
- Since the 2018/19 academic year, a *MATLAB Campus* license has been available for the University of Trieste, allowing teachers and students to download and use MATLAB®, and the associated toolboxes, on their laptops.
- Information on installation methods is available on the website of the Department of Engineering and Architecture (DIA), at the address: `https://dia.units.it/it/didattica/node/32619`
- By registering you can also access the (free) online MATLAB courses, among which we recommend, in order to easily solve the assigned exercises:
  1. *MATLAB Onramp* (very simple, just some basics, approx. 2 hours)
  2. *Make and Manipulate Matrices* (approx. 1,5 hours)
  3. *MATLAB Desktop Tools and Troubleshooting Scripts* (approx. 1 hour)
  4. *Explore Data with MATLAB Plots* (approx. 1,5 hours)
  5. *Programming Constructs* (approx. 1,5 hours)
  6. And many others...

  The first three should be taken *as early as possible*, during the course.
  Then there are specific courses, such as *A Tour of MATLAB Data Types*, *MATLAB Coding Practices for Efficiency and Performance*, *Machine Learning with MATLAB* etc.
- The actual ANSYS CAMPUS licenses are not node-limited (previously limited to 512 knodes).

News AA 2024/25 - *cont.*

Possibility, for one or two teams of interested students, to carry out *3D scans* of objects/systems of interest to then analyse with CFD techniques:

- Using the professional 3D scanner *Rangevision Spectrum* (3DLab):

- Using the *ANSYS Fluent Mesher*:
  - Watertight geometry
  - Fault-tolerant meshing
- Possibility - in future courses - to perform *reverse engineering* (ANSYS SpaceClaim) with subsequent geometric parameterizations and shape optimizations.

Q & A