UNIVERSITÀ
POLITECNICA
DELLE MARCHE

Numerical Heat Transfer
for Applications

Course summary

Dr Valerio D’Alessandro

Contacts: **Ing. Valerio D'Alessandro**

tel: 071 220 4776 email: v.dalessandro@univpm.it

Office hours for students: Wednesday 11:00 – 13:00

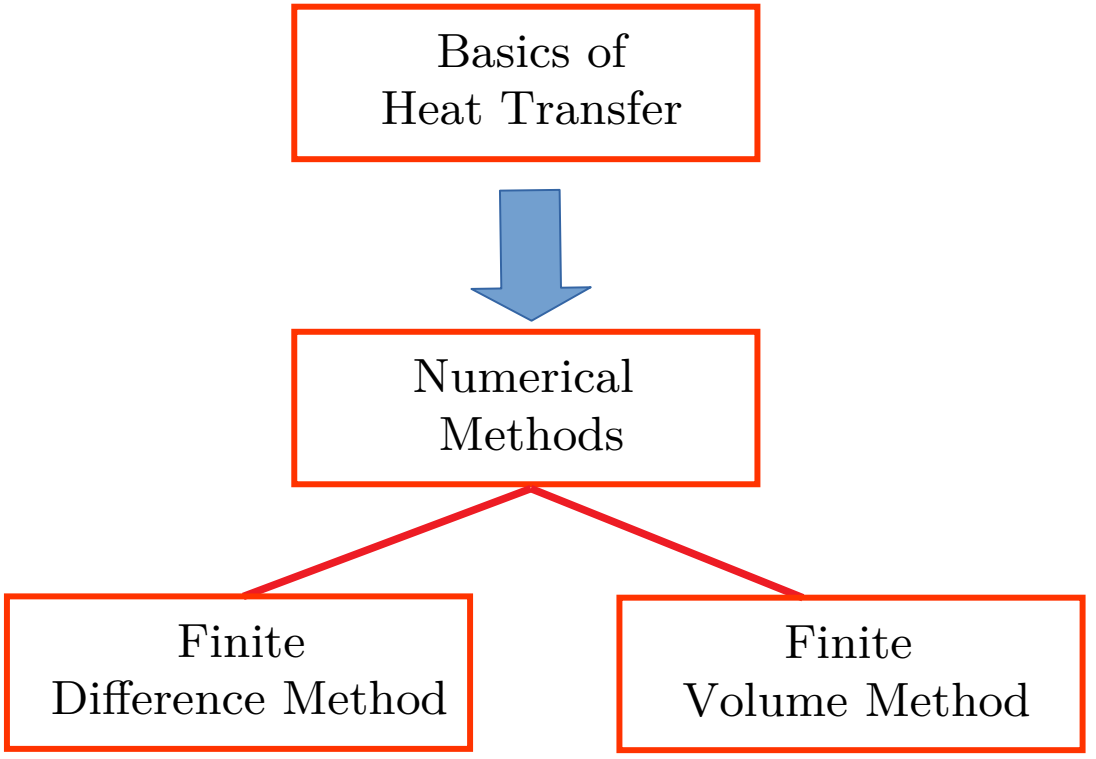
Room: Dipartimento di Ingegneria Industriale e Scienze Matematiche (DIISM) Q160_059

To enter into the Department you have to phone 4776 or 4764 and to require the door opening.

Course collaborator:

Ing. Matteo Falone tel: 071 220 4764 - Q160_071 email: m.falone@staff.univpm.it

Course structure



Course structure

Finite Difference Method

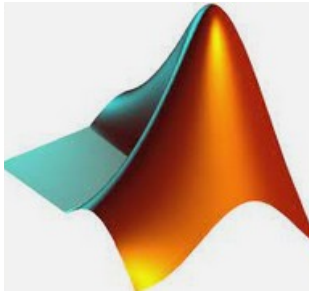
Finite Volume Method

- Finite difference for steady state heat conduction
- Finite volume methods for Laplace equation. Unstructured grids, topological information. Grid geometric quantities. Non-orthogonal unstructured grid. OpenFOAM grid format.
- Fourier equation. Explicit and implicit time integration schemes for finite difference and finite volume methods.
- Introduction to linear systems solution techniques. Jacobi and Gauss-Siedel methods. Tri–diagonal matrix algorithm (Thomas algorithm). Preconditioned conjugate and bi–conjugate gradient methods (hints).
- Applications: thermal modelling of extended surfaces for power electronic applications; bio–heat transfer in skin layer under laser heating;

Course structure

Theory ⟺ Exsercises

Programming environment

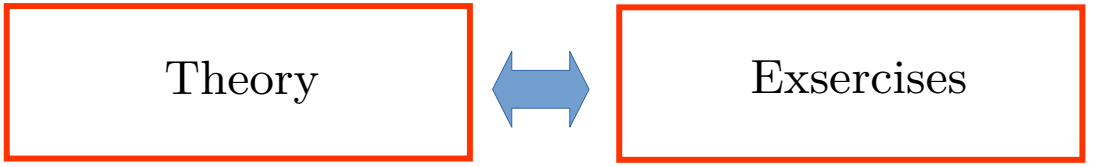


Post-processing

tecplot

ParaView

Course structure



- **MATLAB/Octave** is the development platform.
- For UNIVPM affiliated PhD students, it is important to remark that we have an academic license for all the people which are affiliated to our university.
- Octave is open-source.
- For finite volume methods exsercises is important to download readOpenFoamMesh.m and wrtfld.m routines from https://learn.univpm.it/
- For finite volume methods exsercises post-processing can be performed using **Tecplot** or **Paraview**. Tecplot is a licensed software (in many research group is avaiable), while Paraview is open-source.

Further readings

- Darwish et al. .The Finite Volume Method in Computational Fluid Dynamics (uFVM is used in this course as starting point to read to OpenFOAM mesh files)

- H. Weller and C. Greenshield. Notes on Computational Fluid Dynamics: General Principles

- H. Jasak, PhD Thesis. The Finite Volume Method in Computational Fluid Dynamics