Finite Volume Method (FVM) – DJERAD Abdelkader

Finite Volume Method

Course handouts

Finite Volume Method – Lecture Notes

University	University of M'Sila-Mohamed Boudiaf
Faculty / Institute	Faculty of Technology
Department	Mechanical Engineering
Level / Major	Academic Master / Energetics
Academic year	2025-2026
Professor	Dr. DJERAD ABDELKADER

2026

Foreword

This document brings together the essential concepts of the finite volume method (FV) as used in heat transfer and computational fluid dynamics (CFD). It was designed in the form of a handout of courses and revision sheets: definitions, key formulas, discretization process, and resolution algorithms.

Educational objectives

At the end of the module, the student should be able to:

- write a conservation equation in integral form and identify the flows.
- construct the discretization by control volumes in 1D and 2D.
- choose a suitable convection-diffusion scheme (CDS, UDS, hybrid, TVD/MUSCL).
- Understand pressure-velocity coupling and apply SIMPLE/SIMPLER/PISO.
- Efficiently solve the resulting algebraic system (TDMA, iterative methods).
- Treat unsteady problems and analyze stability.

Educational objectives

At the end of the module, the student should be able to:

- write a conservation equation in integral form and identify the flows.
- construct the discretization by control volumes in 1D and 2D.
- choose a suitable convection-diffusion scheme (CDS, UDS, hybrid, TVD/MUSCL).
- Understand pressure-velocity coupling and apply SIMPLE/SIMPLER/PISO.

- Efficiently solve the resulting algebraic system (TDMA, iterative methods).
- Treat unsteady problems and analyze stability.

Recommended Prerequisites

- Basic numerical analysis: Taylor series, approximation of derivatives.
- Partial differential equations: notions of elliptical/parabolic/hyperbolic.
- Linear algebra: linear systems, sparse matrices, iterative methods.
- Fluid Mechanics and Heat Transfer (if the course is CFD/energy oriented).

Conventions and notations

Golden Rule (MVF)

The MVF is a conservative method: the sum of the outflows of a control volume equals the variation of the quantity conserved in this volume (to the source).

This property remains true at the discrete level, making it particularly robust for flows and transfer.

Throughout the document, the following convention is adopted: a cell (control volume) is denoted P, its neighbors E (East), W (West), N (North), S (South), and the corresponding faces are e, w, n, s. Convective and diffusive fluxes are evaluated at the faces.

Table of Contents

Tip: In Microsoft Word, right-click on the table of contents and then "Update field" to display the pages.

Contents (print version)

- Higher-Order Schemes: QUICK, TVD, and MUSCL
- Digital effects: scattering and scattering
- Example: Stationary 1D Convection-Diffusion

Chapter 4 – SIMPLE / SIMPLER / PISO Algorithms

- Pressure–Speed Coupling
- Maillages staggered et collocated
- Prince of SIMPLE
- SIMPLER
- PISO
- Convergence and controls

Chapter 5 – Solution of Algebraic Equations

- Structure of the discrete system
- TDMA
- Basic Iterative Methods
- Krylov methods
- Multigrid
- Stopping Criteria

Chapter 6 – Transient Problems

- Unsteady term in MVF
- Time Patterns
- Stability (Fourier, CFL)
- Procedure for a time step
- Example: Unsteady Conduction
- Aperture: Space–Time Finite Volumes

Chapter 7 – ψ–ω (2D) formulation

- Motivation
- Definitions

Chapter 1

Chapter 1

Chapter 1: General information on computational fluid dynamics (CFD)

Why CFD?

CFD (Computational Fluid Dynamics) is a group of numerical methods that make it possible to simulate flows and transfers (momentum, heat, chemical species). In practice, CFD complements experimentation: it makes it possible to explore conditions that are difficult to measure, to visualize internal fields (velocity, pressure, temperature) and to optimize geometries.

- Typical applications: exchangers, pipelines, turbomachinery, combustion, aerodynamics, microfluidics.
- The objective: to solve PDEs resulting from conservation laws, with realistic boundary conditions.

Conservation laws: integral form and general equation

The conceptual basis of the MVF is the writing of the balance sheets in integral form on a control volume V delimited by a surface ∂V. For a scalar quantity φ (or component of a vector), the most general conservation equation is written:

$$\frac{\mathbf{d}}{\mathbf{d}t}\int_V \rho\varphi \mathbf{d}V + \int_{\partial V} \rho\varphi(\mathbf{u}\cdot\mathbf{n})\mathbf{d}A = \int_{\partial V} \Gamma\nabla\varphi\cdot\mathbf{n}\mathbf{d}A + \int_V S_\varphi \mathbf{d}V \qquad (1)$$

The three mechanisms are: (i) temporal accumulation, (ii) convection (transport by the u-flow), (iii) diffusion (gradient transport) and (iv) sources/volume sinks.

Examples of φ choices

- Continuity : $\varphi = 1 \rightarrow \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathrm{u}) = 0$
- Momentum: $\varphi = u_i \rightarrow$ Navier–Stokes equations
- Energy / enthalpy: $\varphi = h \rightarrow$ energy equation (with thermal diffusion term).

Classification of PDEs and Impact on Schematics

In numerical analysis, the nature of the PDE (elliptical, parabolic, hyperbolic) guides the choice of schemes (stability, propagation, boundary conditions).

Type	Example	Comments (very summarized)
Elliptical	$\nabla^2 \varphi = S$ (Poisson/Laplace)	Stationary problems; "global" information; Iterative solvers.
Parabolic	$\frac{\partial \varphi}{\partial t} = \alpha \nabla^2 \varphi$ (diffusion instat.)	Temporal stability; explicit/implicit schemas.
Hyperbolic	$\frac{\partial \varphi}{\partial t} + \mathrm{u} \cdot \nabla \varphi = 0$ (advection)	Wave/edge propagation; upstream/TVD schemes, CFL.

CFD Simulation Chain (Overview)

- Pre-processing: geometry, mesh (structured or not), choice of physical model.
- Discretization: choice of method (MVF), spatial and temporal schemes, interpolation to faces.
- Resolution: iterative solver, pressure-velocity coupling, convergence criteria.
- Post-processing: fields, profiles, overall balances (flow, pressure drops, heat flux).

- Verification/validation: mesh independence, comparison to a reference solution or experience.

Mesh independence (minimum)

Redo the simulation on at least 2-3 meshes (coarse → fine) and check that the quantities of interest (flow, Nusselt, pressure drop) vary little.

Reducing the residue is necessary, but not sufficient: also monitor the balances (mass/energy).

Why FVD?

Compared to finite differences (MDF) and finite element (FEM), MVF is very popular in CFD because it naturally conserves flows across the faces of the control volumes, and it adapts well to complex meshes (polyhedral, triangular, etc.).

- Local and global conservation of quantities (at the discrete level).
- Clear physical interpretation: each term corresponds to a flow through a face.
- Easy to extend: convection, diffusion, sources, unsteady terms.

Mini-exercise (review)

For an air flow ($\rho = 1.2$ kg/m^3, $\mu = 1.8\times10^{-5}$ Pa·s) in a channel of height L = 0.05 m with an average velocity U = 2 m/s, calculate the Reynolds number $\mathrm{Re} = \frac{\rho U L}{\mu}$ and discuss whether the flow is probably laminar or turbulent (order of magnitude).

Chapter 2

Chapter 2

Chapter 2: The finite volume method for diffusion problems

Diffusion model and thermal example

Diffusion phenomena describe the transport of a quantity φ under the effect of a gradient (temperature, concentration, potential). In steady-state mode, a generic model is:

$$\nabla \cdot (\Gamma \nabla \varphi) = S_{\varphi} \tag{2}$$

In heat transfer by conduction (isotropic medium), we find: $\nabla \cdot (k \nabla T) + q''' = 0$, where k is the thermal conductivity and q''' is a volume generation.

Discretization by control volumes (principle)

- Divide the domain into non-overlapping control volumes.
- Integrate the differential equation on each control volume.
- Transform volume integrals into surface flux (Gauss's theorem).
- Approximate the flows to the faces using a profile/interpolation.

The following figure sets out the 1D notation used throughout the chapter.

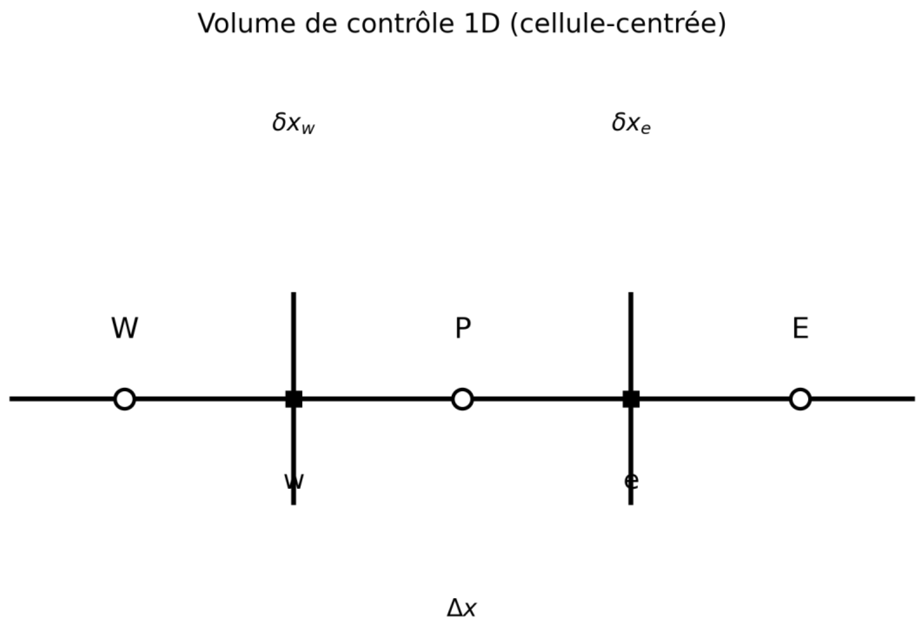

Figure 2.1 – 1D notation (cell-centered): W–P–E nodes and w–e faces.

Case 1D: stationary diffusion (non-uniform mesh)

Consider equation 1D: $\frac{\mathrm{d}}{\mathrm{d}x}\left(k\frac{\mathrm{d}T}{\mathrm{d}x}\right) + S = 0$. After integration on the control volume of P between the w and e faces:

$$\left[k\frac{\mathrm{d}T}{\mathrm{d}x}\right]_e - \left[k\frac{\mathrm{d}T}{\mathrm{d}x}\right] + \overline{S}\,\Delta x = 0_w \tag{3}$$

Assuming a piecewise linear profile between the cell centers, we approximate the diffusive fluxes:

$$\left[k\frac{\mathrm{d}T}{\mathrm{d}x}\right]_e \approx k_e\frac{T_E - T_P}{\delta x_e}\,; \tag{4}$$

$$\left[k\frac{\mathrm{d}T}{\mathrm{d}x}\right]_w \approx k_w\frac{T_P - T_W}{\mathrm{d}x_w}$$

We obtain the standard algebraic equation:

$$a_P T_P = a_E T_E + a_W T_W + b \quad (5)$$

with $a_E = \frac{k_e}{\delta x_e}$, $a_W = \frac{k_w}{\delta x_w}$, $a_P = a_E + a_W - S_P$, and $b = S_C$ + edge terms (if needed).

Treatment of the source term (linearization)

To ensure a physically consistent solution and diagonal dominance, the source term is often linearized as:

$$\overline{S} = S_C + S_P T_P \ (\text{with } S_P \leq 0) \quad (6)$$

Why impose S_P ≤ 0?

This condition helps to maintain "positive" coefficients in the sense of MVF (bornitude) and avoids the appearance of non-physical numerical solutions.

Equivalent Conductivity at Interfaces

If the conductivity varies from one cell to another (different materials), the best practice is to evaluate k_e by a harmonic mean, which corresponds to the addition of the thermal resistances:

$$k_e = \frac{1}{\frac{f_e}{k_P} + \frac{(1 - f_e)}{k_E}} \quad (7)$$

(if $f_e = 1/2 \Rightarrow k_e = \frac{2k_P k_E}{k_P + k_E}$)

This average correctly respects the limit cases (insulating wall, strong contrast of k).

Boundary conditions (Dirichlet, Neumann, Robin)

Boundary conditions are naturally dealt with by integrating the equation over a half-volume of control adjacent to the boundary.

- Dirichlet: T imposed → the boundary value is known.
- Neumann: imposed flux $q_B = -k\frac{dT}{dn}$.
- Robin : convection $q_B = h(T_\infty - T_B)$.

The result is an equation of the type $a_B T_B = a_E T_E + b$, where a_B and b integrate the imposed flux or the convective exchange.

Assembly and Resolution: Tridiagonal System (TDMA / Thomas)

In 1D, the assembly on N nodes produces a tridiagonal matrix. The reference solver is the Thomas algorithm (TDMA), equivalent to a specialized Gaussian elimination.

- Forward scanning: calculation of the modified coefficients P_i and Q_i.
- Rear scan: calculation of T_N and then up to T_1.

> **Good reflex**
>
> Always check: (i) neighbor coefficients ≥ 0, (ii) a_P ≥ sum of neighbors (diagonal dominance), (iii) coherent overall balance (inflow/outflow).

Guided Example (1D)

Consider a plane wall $0 \leq x \leq L$, k constant, generation q''' uniform, T(0)=T_0 and T(L)=T_L. Construct the MVF discretization with N cells, write the coefficients a_W, a_E, a_P and the second member b. Then compare the numerical solution to the analytical solution (quadratic profile).

Chapter 3

Chapter 3

Chapter 3: The finite volume method for convection-diffusion problems

Convection-diffusion equation (integral form)

For a scalar φ transported by a field of velocity u, with Γ scattering and S_φ source:

$$\frac{\partial(\rho\varphi)}{\partial t} + \nabla\cdot(\rho \mathrm{u}\varphi) = \nabla\cdot(\Gamma\nabla\varphi) + S_\varphi \tag{8}$$

The MVF leads to a flow balance across the faces: convective flux $F_f = \rho(\mathrm{u}\cdot\mathrm{n})_f A_f$, diffusive flux $D_f = \frac{\Gamma_f A_f}{d_f}$.

Péclet number and convection/diffusion competition

Pe = $\frac{\rho U L}{\Gamma}$ (or locally $\mathrm{Pe}_f = \frac{F_f}{D_f}$)

- Pe ≪ 1: dominant diffusion (smooth profile) → effective centered schemes.
- Pe ≫ 1: dominant convection (steep fronts) → risk of oscillations if the pattern is inadequate.

1D discretization: form of coefficients

In stationary 1D, the discrete equation is always written as: $a_P\varphi_P = a_E\varphi_E + a_W\varphi_W + b$. The main difference with pure diffusion is the calculation of φ to faces (φ_e, φ_w), which depends on the convective pattern.

Classical schemes (CDS, UDS, hybrid)

Diagram	Order	Benefits	Cons
CDS (centered)	2	Accurate if low Pe	Oscillations if high Pe (unbounded)

UDS (upstream)	1	Stable and stubborn	Digital distribution (too "spread out" profile)
Hybrid	1–2	Combined CDS/UDS according to Pe	Non-smooth transition; depends on a criterion

Higher-Order Schemes: QUICK, TVD, and MUSCL

Higher-order schemes reduce digital diffusion but must remain "bounded" (not create new extrema). The TVD/MUSCL approaches introduce a limiter φ(r) based on the ratio of r-gradients.

$$\varphi_f = \varphi_{\mathrm{up}} + 0.5\varphi(r)(\varphi_D - \varphi_{\mathrm{up}})\,; \quad r = \frac{\varphi_{\mathrm{up}} - \varphi_{\mathrm{UU}}}{\varphi_D - \varphi_{\mathrm{up}}} \tag{9}$$

Examples of limiters: minmod, van Leer, superbee. The choice depends on the compromise between precision and robustness.

> **Point important**
>
> When the field presents discontinuities or very steep gradients (shocks, fronts), favor
>
> bounded patterns (TVD) to avoid non-physical oscillations.

Digital effects: scattering and scattering

- Digital diffusion: upstream "spreads" the fronts (dissipative error).
- Digital dispersion: Some high-order patterns create oscillations (dispersive error).

- The selection of the scheme depends on Pe, the mesh, and the physics (edges vs. smooth profiles).

Example: Stationary 1D Convection-Diffusion

For a domain $0 \leq x \leq L$ with velocity U>0, constant Γ, and conditions $\varphi(0) = \varphi_0, \varphi(L) = \varphi_L$, the analytical solution shows a boundary layer when Pe is large. A classic exercise consists of comparing CDS and UDS for different Pe and observing the appearance of oscillations (CDS) or digital diffusion (UDS).

Chapter 4

Chapter 4

Chapter 4: Solution algorithms (SIMPLE, SIMPLER, PISO)

Pressure–velocity coupling problem

For incompressible flows (constant ρ), continuity requires: $\nabla \cdot \mathrm{u} = 0$. However, pressure does not have its own equation of evolution; it appears as a Lagrange multiplier ensuring the zero divergence constraint.

The SIMPLE/SIMPLER/PISO algorithms construct a pressure correction equation from continuity and momentum equations.

Maillages décalés (staggered) et collocated

- Staggered: you and v are stored at the faces, p in the center → natural coupling, few oscillations.
- Collocated: all variables in the center → simplest on unstructured meshes, but requires special interpolation (Rhie–Chow type) to avoid the pressure "checkerboard".

Principle of SIMPLE (Steady / pseudo-transient)

Idea: start from a pressure assumed p^*; solve the momentum equations to obtain an intermediate velocity u^*; then correct pressure and speed to satisfy continuity.

1. Initialize p^* (and possibly u^*).

2. Solve the momentum equations → u^* using p^*.

3. Construct the pressure correction equation p' from the continuity.

4. Update: $p = p^* + \alpha_p p'$; $\mathrm{u} = \mathrm{u}^* + \mathrm{u}'$ (α_p: relaxation).

Sub-relaxation

In strongly nonlinear problems, the sub-relaxation ($0<\alpha<1$) stabilizes the iteration. Too weak: slow convergence; too high: possible divergence.

Repeat until convergence (residues and balances).

SIMPLER (improvement of SIMPLE)

SIMPLER introduces an additional step: an "enhanced" pressure equation is solved before correction, which reduces the sensitivity to sub-relaxation and often accelerates steady-state convergence.

PISO (Transient)

PISO is particularly suitable for unsteady flows: at each time step, it makes several pressure–velocity corrections (without strong under-relaxation) to achieve a near-zero velocity field at the same time step.

- 1 prediction + 2 (or more) pressure corrections per time step.
- Higher cost/iteration, but fewer overall iterations and better time accuracy.

Convergence criteria and controls

- Normalized residuals (momentum, continuity, energy).
- Mass balance: incoming ≈ outgoing flow (low relative error).
- Stabilization of integral quantities: forces, pressure drops, Nusselt, etc.

Algorithm	Typical use	Advantages	Limitations
SIMPLE	Stationary	Robust, widespread	Can be slow; Requires relaxation
SIMPLER	Stationary	Convergence often faster	Somewhat longer implementation
PISO	Unsteady	Good time accuracy	Several corrections per step

Chapter 5

Chapter 5

Chapter 5: Solution of discretized algebraic equations

Structure of the discrete problem

MVF discretization produces a linear (or non-linear) system of large size:

$$A\,x = b \tag{10}$$

where A is sparse. The solution strategy depends strongly on: the dimension (1D/2D/3D), the number of unknowns, and the conditioning of the system.

Reminder

In pure 1D diffusion, A is tridiagonal → TDMA is very efficient.

In 2D/3D, A is hollow with a stencil (5 points, 7 points, or more) → iterative methods.

Direct Methods and TDMA

The most widely used direct method in 1D is TDMA (Thomas). In 2D, TDMA can be applied line-by-line, but convergence depends on the scan order and coupling.

Basic Iterative Methods

Method	Principle	Benefits	Notes
Jacobi	Simultaneous update	Single	Slow convergence; Useful for preconditioner
Gauss-Seidel	Sequential update	Faster than Jacobi	Depends on the scan
SOR	Gauss-Seidel + relaxation	Can accelerate strongly	Optimal choice ω delicate

Krylov methods (hollow systems)

For large systems derived from CFDs, Krylov methods (CG, BiCGStab, GMRES) are very efficient, provided that preconditioning is used (Jacobi, ILU, AMG, etc.).

- CG: for positive defined symmetric matrices (pure diffusion).
- BiCGStab / GMRES: for non-symmetric matrices (convection-diffusion, Navier–Stokes).
- Preconditioning improves packaging and reduces the number of iterations.

Multigrid (principle)

Multigrid accelerates convergence by handling errors at different scales: a straightener (Gauss–Seidel) eliminates high frequencies, then coarse mesh correction eliminates low frequencies (V or W cycle).

Error Measurement and Shutdown

- Residue $r = b - Ax$: follow $\|r\|$ (L2 or max).
- Criterion on the relative variation of x (stagnation).
- Physical control: balances (mass, energy) and integral quantities.

Chapter 6

Chapter 6

Chapter 6: The finite volume method for transient flows

Unsteady term in FVM

Unsteadiness translates into an accumulation term. By integrating on a V_P control volume:

$$\int_V \frac{\partial(\rho\varphi)}{\partial t}\mathrm{d}V \approx (\rho_P V_P)\,\frac{{\varphi_P}^{n+1} - {\varphi_P}^{n}}{\Delta t} \tag{11}$$

At the discrete level, the unsteady term behaves as an additional contribution on the diagonal (augmented a_P) and a state-dependent source term at the previous step.

Time Patterns (θ-Method)

A temporal interpolation of the flows according to a parameter θ (0≤θ≤1) is often used:

$$\varphi^{n+1} = \varphi^{n} + \Delta t\,[\theta R(\varphi^{n+1}) + (1 - \theta)R(\varphi^{n})] \tag{12}$$

- θ=0: explicit (Euler before) – simple but conditionally stable.
- θ=1: implicit (rear Euler) – stable (diffusion) but less precise.
- θ=1/2: Crank–Nicolson – order 2 in time, but can oscillate if not damped.

Stability: Fourier numbers and CFL

- Diffusion (explicit): $\mathrm{Fo} = \frac{\alpha\Delta t}{\Delta x^2}$ must remain small (depending on the size).
- Convection (explicit): $\mathrm{CFL} = \frac{|u|\Delta t}{\Delta x} \leq 1$ (order of magnitude).
- Implicitly, the constraint is less severe, but Δt influences the precision.

Typical Procedure for a Time Step

- Update time-dependent terms (sources, properties).
- Assemble the discrete equations at time n+1 (according to θ).
- Solve the system (often non-linear → internal iterations).
- Possibly apply PISO (several pressure–velocity corrections).
- Move on to the next step and save the useful fields.

Example: Unsteady 1D Conduction

Equation:

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x}\right) \tag{13}$$

The MVF leads to a relation of the type:

$$a_P {T_P}^{n+1} = a_E {T_E}^{n+1} + a_W {T_W}^{n+1} + b \tag{14}$$

where b contains the contribution $\frac{\rho c V}{\Delta t} {T_P}^n$.

Aperture: Space–Time Finite Volumes (General Idea)

For some problems (moving boundaries, phase change), we can build control volumes in space and time simultaneously (space–time approach), or use a moving mesh. The aim is to preserve conservation even as geometry changes.

Chapter 7

Chapter 7

Chapter 7: ψ–ω Method for Convection-Diffusion Problems (2D Flow)

Motivation

In incompressible 2D, the vorticity–current function (ω–ψ) formulation eliminates pressure. It is particularly useful for teaching and for simple geometries (cavity, canal), because it reduces the problem to two coupled scalar equations.

Definitions

$$u = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x} \quad \omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \tag{15}$$

With these definitions, the continuity $\nabla \cdot \mathrm{u} = 0$ is automatically satisfied.

Governing equations (2D)

$$\frac{\partial \omega}{\partial t} + u\frac{\partial \omega}{\partial x} + v\frac{\partial \omega}{\partial y} = \nu \nabla^2 \omega \tag{16}$$

$$\nabla^2 \psi = -\omega \tag{17}$$

The first equation is a convection-diffusion (see Chapter 3), the second is a Poisson (see Chapter 2).

Boundary Conditions

- No-slip (fixed walls): u = v = 0.
- ψ is constant on a closed wall (often ψ=0).
- ω at the wall is obtained by a discrete relation involving ψ (Laplacian approximation near the edge).

Calculation algorithm (simple scheme)

- Initialize ω (and ψ).
- Solve $\nabla^2\psi = -\omega$ to obtain ψ.
- Calculate you and v from ψ.
- Solve the transport equation of ω (convection-diffusion) to obtain ω.
- Repeat until convergence (stationary) or advance in time (unsteady).

Reference Example: Lid-Driven Cavity

Standard problem: a square cavity with the top cover moving at a constant speed U. The formation of a main vortex and secondary vortices according to the Reynolds number is observed.

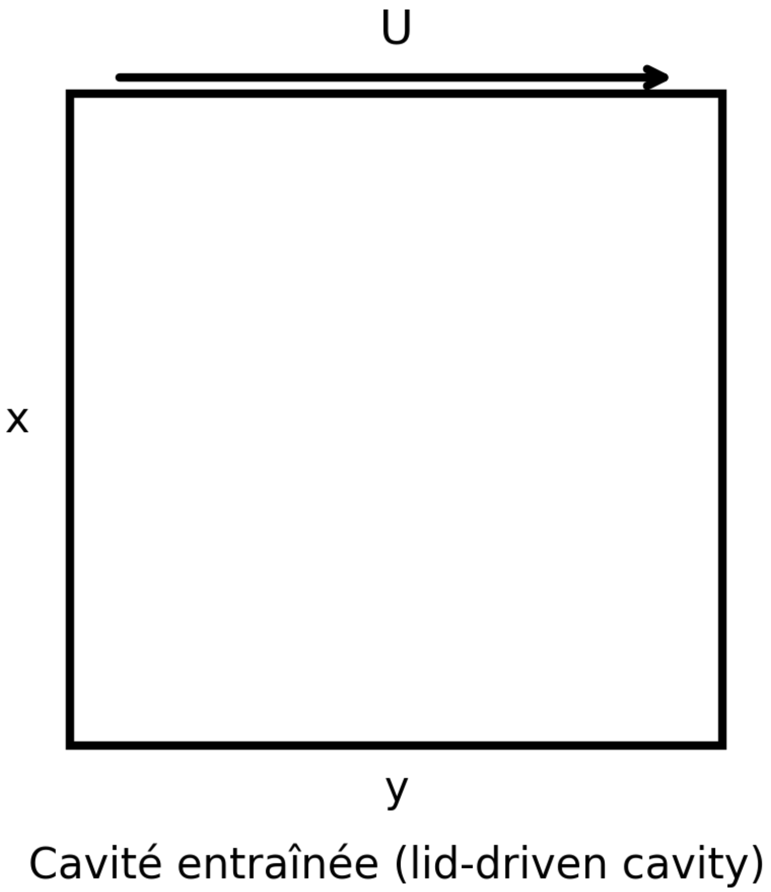

Figure 7.1 – Driven cavity: moving upper wall.

Limitations and Extensions

- Mainly 2D; the 3D extension is not direct.

- Difficult complex geometries without transformation or adapted mesh.
- Possible extensions: thermal coupling, turbulence models (RANS), axisymmetric formulation.

Conclusion

The finite volume method is one of the most robust frameworks for numerical simulation of transfers and flows today. Its main advantage is the discrete level conservation, which results in reliable overall balances. The choice of a schematic (centered, upstream, TVD), an algorithm (SIMPLE/PISO) and a suitable linear solver is decisive to obtain an accurate and robust solution.

Quick checklist before validating a calculation

- Sufficiently low and stabilized residues.
- Mass balance respected (low relative error).
- Mesh independence tested (at least 2 refinements).
- Controlled (transient) time-step sensitivity.

Bibliographical references (selection)

- Patankar, S.V. Numerical Heat Transfer and Fluid Flow. Hemisphere, 1980.
- Versteeg, H.K. & Malalasekera, W. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson, 2e éd., 2007.
- Ferziger, J.H. & Perić, M. Computational Methods for Fluid Dynamics. Springer, 3e éd., 2002.
- LeVeque, R.J. Finite Volume Methods for Hyperbolic Problems. Cambridge University Press, 2002.
- Moukalled, F., Mangani, L., Darwish, M. The Finite Volume Method in Computational Fluid Dynamics. Springer, 2016.
- Derradji, S. Finite Volumes – Course summary, tutorial and practical work (Master 1 Energetics). Batna 2 University, 2019/2020.
- Vovelle, J. Introduction to finite volume methods (lecture notes / teaching paper).
- Perron, J.C. A finite volume method deformable in space-time for phase change in compressible media.
- Aboshighiba, H. Advanced Numerical Methods – Handout of courses with exercises. 2021–2022.
- Kaid, N. The finite volume method for diffusion problems (lecture notes). Salhi Ahmed Naama University Center, 2016–2017.
- Abaali, M. Finite volume method (ENSA 2024–2025 support).
- Boivin, S. & Hérard, J.-M. A finite volume scheme to solve the Navier–Stokes equations on a triangulation. European Journal of Finite Elements, Vol. 5, No. 4, 1996, pp. 461–490.

Appendices

Appendix A: Key Ratings

Symbol	Meaning
f	Generic scalar variable
u, v, w	Speed Components
p	Pressure
r	Density
m	Dynamic viscosity
n	Viscosité cinématique (ν=μ/r)
C	Diffusion coefficient
S_f	Source term
Pe	Péclet number (convection/diffusion)
Re	Reynolds number (inertia/viscosity)

Appendix B: Qualitative Properties of Convective Patterns

Diagram	Order	Bounded	Conservative	Transportive
UDS (upstream)	1	Yes	Yes	Yes
CDS (centered)	2	No (High Pe)	Yes	No
Hybrid	1–2	Yes	Yes	Yes
QUICK	3	No (no correction)	Yes	Partial

TVD/MUSCL	≥2	Yes	Yes	Yes

Appendix C: Series of Exercises (to be covered)

These exercises are offered for training. They can be treated in tutorials or mini-projects.

- 1D broadcasting: solving $\nabla \cdot (k\nabla T) + q''' = 0$ over [0,L] with T(0)=T0, T(L)=TL (k constant), and compare to analytics.
- 2D scattering: Laplace $\nabla^2 T = 0$ in a square with 3 isothermal edges and 1 isolated edge; implement GS or SOR.
- 1D convection-diffusion: compare UDS and CDS for Pe = 0.1; 2 ; 10 and analyze oscillations/digital diffusion.
- 2D convection-diffusion: uniform flow in a channel; introduce a TVD scheme and compare to UDS.
- SIMPLE: implement a 2D version for the incompressible Poiseuille flow (channel), structured mesh.
- ψ–ω: cavity driven at Re=100; Get the streamlines and locate the main vortex.

Appendix D: 2D FVM on Unstructured Meshes (Dissemination)

In real CFD applications, geometry often requires unstructured meshes (2D triangles, 3D tetrahedra) or polyhedral meshes. The MVF adapts naturally to this because it starts from a complete assessment of each cell.

Consider the stationary scattering equation: $-\nabla \cdot (\kappa \nabla u) = f$ in Ω, with boundary conditions on ∂Ω.

Integration on a K cell (control volume) and application of Gauss's theorem:

$$\sum_{\sigma \in \partial K} F_{K,\sigma} = |K| f_K \,, \tag{18}$$

$$\text{où } F_{K,\sigma} = -\int_{\sigma} \kappa \nabla u \cdot \mathrm{n}_{K,\sigma} \, \mathrm{d}\gamma$$

The core of the scheme is therefore the approximation of the diffusive flux through each face σ.

2-point approximation (TPFA) – orthogonal case: if σ is a common face between two cells K and L, and if the segment joining the centers x_K and x_L is (almost) orthogonal to the face, we can write:

$$F_{K,\sigma} \approx -k_{K,\sigma} \, |\sigma| \, \frac{u_L - u_K}{d_{K,L}} \tag{19}$$

where $|\sigma|$ is the length (2D) of the face, and $d_{K,L}$ is the distance between centers measured in the direction normal to σ.

Edge faces:

• Dirichlet (imposed u): treat the face as a fictitious neighbor of u_σ (or u_B) value and use a distance $d_{K,\sigma}$.

• Neumann (flux imposé) : poser directement $F_{K,\sigma} = |\sigma| \, q_p$.

Important note (orthogonality): TPFA is rigorous on orthogonal meshes (e.g. Voronoi meshes). On non-orthogonal meshes, correction (orthogonal/non-orthogonal decomposition) or a multi-point approach (MPFA) may be necessary to maintain accuracy and robustness.

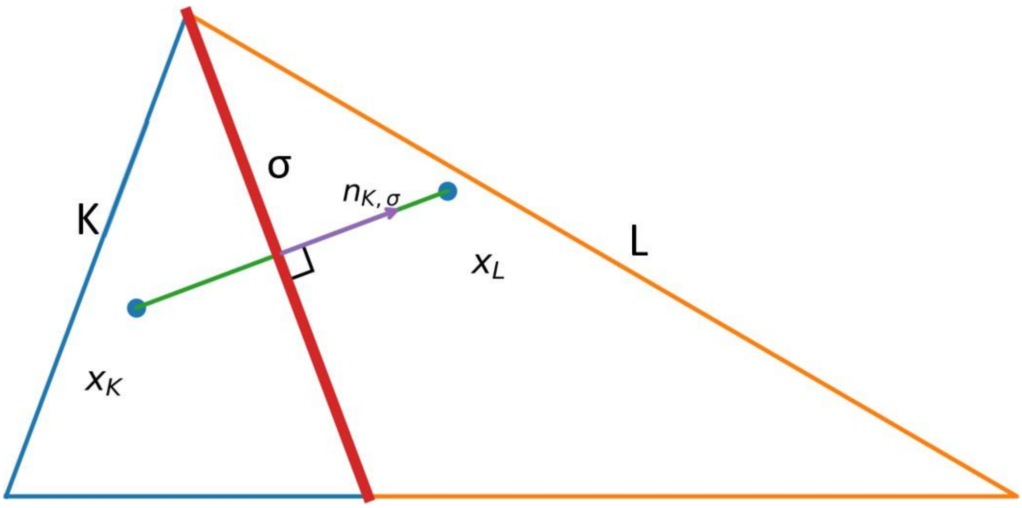

Figure D.1 – Example of two cells (K, L) sharing a σ face: x_K and x_L centers, and normal n_{K,σ}.

Appendix E: Numerical Examples (1D Broadcast) – Assembly and Solution

Example E.1 – Stationary conduction in a rod (Dirichlet conditions)

Data:

L = 0.5 m; A = 10×10^{-3} m^2; k = 1000 W/m.K; $T(0) = T_A = 100$ °C; $T(L) = T_B = 500$ °C.

The rod is divided into N = 5 equal control volumes (Δx = 0.1 m). The unknowns are the temperatures in the center of the cells: $T_1 ... T_5$.

Constant and not uniform conductivity: $a_W = a_E = \frac{kA}{\Delta x} = 100$ (for the internal nodes).

The Dirichlet conditions are taken into account by half-volume at the borders. The resulting discrete system is written in tridiagonal form:

$$300T_1 - 100T_2 = 200T_A$$

$$-100T_1 + 200T_2 - 100T_3 = 0$$

$$-100T_2 + 200T_3 - 100T_4 = 0$$

$$-100T_3 + 200T_4 - 100T_5 = 0$$

$$-100T_4 + 300T_5 = 200T_B$$

Solution (TDMA ou Gauss) : $[T_1, T_2, T_3, T_4, T_5]^T$ = $[140, 220, 300, 380, 460]^T$ (°C).

Verification: the exact solution is linear; these values correspond to T(x)=100+800x at the centers (x=0.05; 0.15; ... ; 0.45 m).

Summary of the coefficients (form $a_P T_P = a_W T_W + a_E T_E + S_U$, with S_P embedded in a_P):

Knot	a_W	a_E	S_U	S_P	a_P
1	0	100	20000	−200	300
2	100	100	0	0	200
3	100	100	0	0	200
4	100	100	0	0	200
5	100	0	100000	−200	300

Note: here $S_U = \left(\frac{2kA}{\Delta x}\right) \cdot T_{\text{frontière}}$ for cells at the edges (half volume).

Example E.2 – 1D stationary scattering with uniform volume source

Plate thickness L = 2 cm (0.02 m) with uniform generation

$q = 1000\ \text{kW/m}^3$ (i.e. $1 \times 10^6\ \text{W/m}^3$) and conductivity k = 0.5 W/m.K. The faces are at $T_A = 100\ °\text{C}$ and $T_B = 200\ °\text{C}$. We assume a 1D problem (significant variations only according to x).

Discretization: N = 5 volumes, Δx = 0.004 m. To simplify, we take a section

A = 1 m². Internal coefficients: $a_W = a_E = \frac{kA}{\Delta x} = 125$; source:

$S_U = qA\Delta x = 4000.$

Coefficients at the edge (Dirichlet, half-volume): appearance of an additional term $\frac{2kA}{\Delta x} = 250$ in S_U and $S_P = -250.$

Solution : $[T_1, T_2, T_3, T_4, T_5]^T = [150, 218, 254, 258, 230]^T$ (°C).

Summary of coefficients:

Knot	a_W	a_E	S_U	S_P	a_P
1	0	125	29000	−250	375
2	125	125	4000	0	250
3	125	125	4000	0	250
4	125	125	4000	0	250
5	125	0	54000	−250	375

Interpretation: the presence of the source term creates a nonlinear profile with a maximum temperature within the domain.

Appendix F: FVD Robustness Checklist (Practice Rules)

This checklist summarizes practical rules for avoiding non-physical numerical solutions (oscillations, negative values, divergence) when constructing an MVF schema.

- Consistency with common faces: for a face shared by two cells, the numerical stream must be written identically in both equations (which guarantees local conservation: what comes out of K enters L).
- Positive neighboring coefficients: in the discretized equation $a_P \varphi_P = \sum a_{\mathrm{nb}} \varphi_{\mathrm{nb}} + b$, it is recommended to have $a_{\mathrm{nb}} \geq 0$. This promotes monotony (no overshoot/undershoot) and a more stable convergence.
- Linearization of sources: if we write $\overline{S} = S_C + S_P \varphi_P$, imposing $S_P \leq 0$ allows us to keep a diagonal term a_P sufficiently large (diagonal dominance) and to avoid instabilities.
- Accuracy for a uniform field: when the PDE is satisfied by φ and $\varphi + c$ (constant c), check that, excluding on-board contributions, $a_P \approx \sum a_{\mathrm{nb}}$ (the sum of the flows cancels out for a constant field).
- Control of the overall balance: the sum of the discrete equations over the entire domain must give a coherent overall balance (inflows/outflows = sources). It is a simple test to detect an assembly error.
- Transient: In an implicit scheme, the time term adds $\left(\frac{\rho V}{\Delta t}\right)$ to a_P, which improves stability. In explicit, check the stability criteria (Fourier/Pe, CFL depending on the problem).

— End of Booklet —

Dr. Abdelkader Djerad | Department of Mechanical Engineering | University of M'Sila