Methods for the Domain Discretization
Focus: Square Cylinder Meshing

Miquel Altadill Llasat

Master Thesis

Supervisors:
M.Sc Charalampos Alexopoulos, ITLR
Prof. Dr.-Ing. B. Weigand, ITLR
Prof. Bassam A. Younis Ph.D, UCD

Universität Stuttgart
Institut für Thermodynamik der Luft- und Raumfahrt
ITLR
UCDAVIS
UNIVERSITY OF CALIFORNIA

30 April 2021

Table of contents

Introduction

Any numerical method developed for the flied field modelling consists on the following ingredients:

- Mathematical model
- Discretization method
- Coordinate and vector system
- **Numerical grid**
- Finite approximations
- Numerical Solver Methods
- Boundary Conditions

The most important discretization methods are:

- Finite difference method (**FDM**)
- Finite volume method(**FVM**)
- Finite element method (**FEM**)

Each type of method yields the same solution if the grid is fine enough.

[NASA image courtesy MODIS Rapid Response Team at NASA GSFC]

Finite Differences Method

Oldest and also the **simplest** discretization method for numerical solutions of PDE's sets.

The usual procedure for deriving finite-difference equations [1] consists of approximating the derivatives in the differential equation using the Taylor-series expansion.

$$\phi_1 = \phi_2 - \Delta x \left(\frac{d\phi}{dx}\right)_2 + \frac{1}{2}(\Delta x)^2 \left(\frac{d^2\phi}{dx^2}\right)_2 - \dots \quad (1)$$

$$\phi_3 = \phi_2 + \Delta x \left(\frac{d\phi}{dx}\right)_2 + \frac{1}{2}(\Delta x)^2 \left(\frac{d^2\phi}{dx^2}\right)_2 + \dots \quad (2)$$

Cutting the series after the third term and after working these equations, getting

$$\left(\frac{d\phi}{dx}\right)_2 = \frac{\phi_3 - \phi_1}{2\Delta x} \quad (3)$$

$$\left(\frac{d\phi}{dx}\right)_2 = \frac{\phi_1 + \phi_3 - 2\phi_2}{(\Delta x)^2} \quad (4)$$

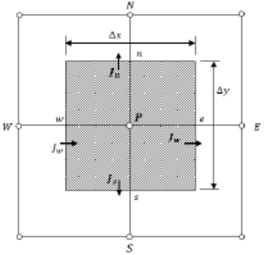
On structured grids, the FDM method is very simple and effective.

Finite Volumes Method

It consists of dividing the the solution domain into a finite number of contiguous CVs. At the centroid of each one lies a computational node where the conservation equations are applied for the problem solving. The integral form of the conservation equation reads as

$$\int_S (\rho \overrightarrow{v} \phi) \cdot n \, dS = \int_S (\Gamma \nabla \phi) \cdot n dS + \int_V G \, dV \tag{5}$$

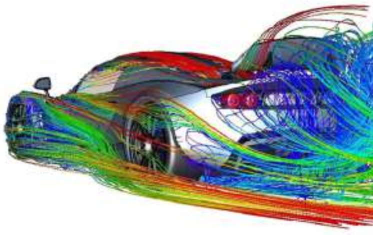
Advantages:
Can accommodate any type of grid
Suitable for complex geometries
Simplicity to program
Physical Meaning
Disadvantages:
Integration difficulties
Laborious implementation into 3D models



Finite Elements Method

The FE-method is similar to the FV-method in many ways. The domain is broken into a set of discrete volumes or finite elements that are generally unstructured.
2D - Triangles or quadrilaterals
3D - tetrahedral or hexahedral

Feature. Equations are multiplied by a weight function before the domain integration.



Advantages:

- Deal with arbitrary geometry
- Grid easily refined

Disadvantages:

- Hard to find efficient solution methods.

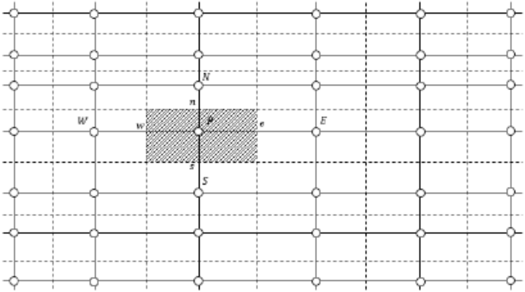
There is a hybrid method called **control-volume-based finite element method** (CVFEM) should be mentioned

Numerical Grid

Focusing on the **FVM**, there are many possible ways for locating the control-volume but for the aim of this thesis only two different grids are explained.

Grid A:

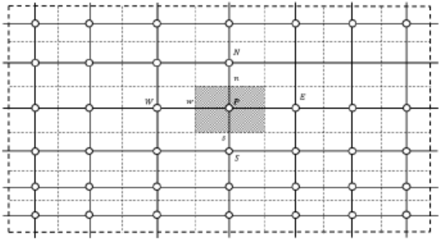
- Place faces midway between neighbouring grid points
- Another observation is that the grid is nonuniform
- P does not lie at the geometric centre of the CV

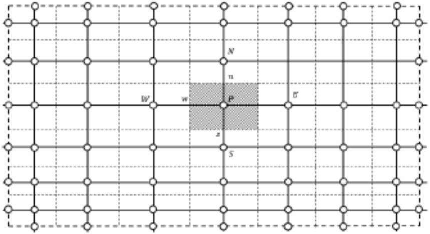


Numerical Grid

Grid B: drawing the control-volume boundaries first and then place the gridpoint at the geometric centre of each control-volume.

It is needed to make additional considerations for the control volume near the boundaries of the domain. It is convenient to place the boundary grid points on the faces of the near-boundary control volume faces.





One of the decisive advantages of Grid B is that the control volume turns out to be the basic unit of the discretization method, it is more convenient drawing the control-volume boundaries first and let the grid-point locations follow as consequence.

N-S Equations

For a better treatment we would use the following mass and momentum equations:

$$\Delta \cdot v = 0 \tag{6}$$

$$\rho \frac{\partial v}{\partial t} + (\rho v \cdot \nabla) v = -\nabla p + \mu \Delta v \tag{7}$$

Pressure - **Velocity** coupling. Due to the close connection of the pressure and the continuity equation, it requires special attention. The pressure distribution allows the velocity field to satisfy the mass conservation equation.

Concepts:

- **Diffusion**. The viscous is the one that depend on the fuild dynamic viscosity, these term is also called diffusive term because of it physical meaning.

- **Convection**. Depend on the fluid velocity. Due to the fluid field movement and the diffusion is produced by the fluid particle interaction due to its inherent vibration.

Fractional Step Method (FSM)

The FSM of Kim and Moin (1985) is more a generic approach than a particular method. Here, the pressure is solved once per time step.

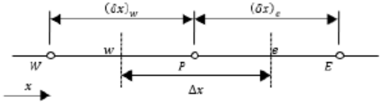
This is largely because fractional step methods are used mainly in unsteady flow simulations while the latter is used predominantly to compute steady flows.

For the simplicity of the current purpose it would only be considered the incompressible formulation of the FSM.

FSM Algorithm

1. Evaluation of $R(v^n)$
2. $v_p = v^n + \frac{\Delta t}{\rho} \cdot \left(\frac{3}{2} R(v^n) - \frac{1}{2} R(v^{n-1}) \right)$
3. $\Delta p^{n+1} = \frac{\rho}{\Delta t} \nabla \cdot v^p$
4. $v^{n+1} = v^p - \frac{\Delta t}{\rho} \nabla p^{n+1}$

The algorithm comes from the application of the Helmholtz-Hodge theorem and the complete procedure can be found in [1] and many other references.



$R(v)$ is function of the convective and diffusive term.

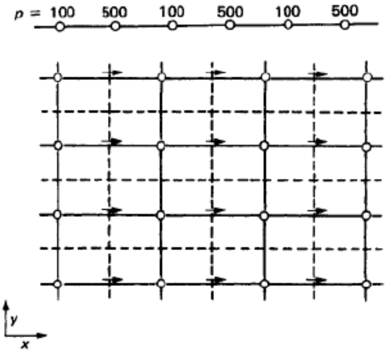
$$R(v) = -(\rho v \cdot \nabla) v + \mu \Delta v \tag{8}$$

Fractional Step Method (FSM)

The explanation of the FSM was necessary in order to introduce the Checkerboard Problem. Discretizing the equation stated in the 4th FSM algorithm step in a 1D domain

A wavy pressure field will be felt like a uniform pressure field by the momentum equation.

$$u_P^{n+1} = u_P^P - \frac{\Delta t}{\rho}\left(\frac{p_E^{n+1} - p_W^{n+1}}{2\Delta x}\right) \tag{9}$$



[1]

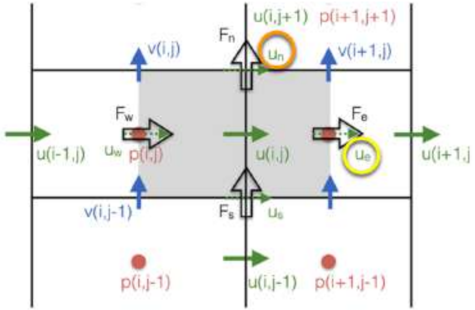
A remedy: Staggered Mesh

- One can employ a different grid for each dependent variable.
- Benefit for arranging the velocity component on grids that are different from the grid used for all other variables

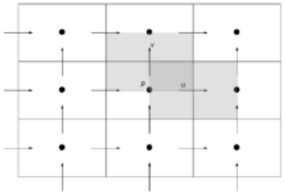
Staggered Mesh

Such a displaced or "staggered" mesh for the velocity components was firstly used by Harlow and Welch (1965) in their MAC method.

In the staggered grid, the velocity components are calculated for the points that lie on the faces of the control volumes. Thus, the x-direction velocity u is calculated at the faces that are normal to the x direction.



[3]



[3]

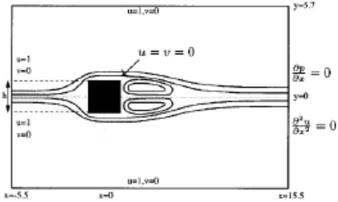
$$u_e = \frac{u_{(i+1,j)} + u_{(i,j)}}{2} \quad (10a)$$

$$u_n = \frac{u_{(i,j+1)} + u_{(i,j)}}{2} \quad (10b)$$

Note that modified Grid B is necessary for the application of this mesh.

Square Cylinder Discretization

Title: DNS OF 2D TURBULENT FLOW AROUND A SQUARE CYLINDER
Author: JAN G. WISSINK —— **Year:** 1997 —— **Re** = 10000



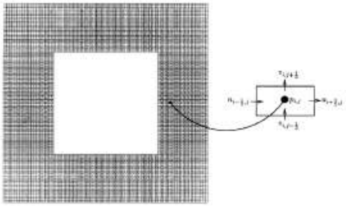


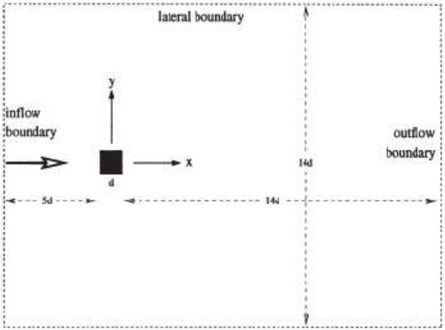
Figure 1. Stretched and staggered grid in neighbourhood of cylinder

Same boundary conditions are applied to the upper boundary and the lower boundary

Square Cylinder Discretization

Title: FLOW PAST A SQUARE CYLINDER: TEST CASE LES2
Author: PETER R. YOKE —— **Year:** 1997 —— **Re** $= 21400$

The domain dimensions of 20d x 14d x 4d. (In fact the groups used a variety of streamwise dimensions for their computational domains downstream of the cylinder, but this variation does not appear to have affected the results in any obvious way.)



Most of the tests used collocated (centered) meshing. NT5 and NT7 used staggered meshing.

Title: SIMULATION OF VORTEX SHEDDING PAST A SQUARE CYLINDER WITH DIFFERENT TURBULENCE MODELS
Author: G. BOSCH1 AND W. RODI —— **Year:** 1998 —— **Re** = 22000

Three different types of computational meshes will be used, namely Frankes' mesh, the 'optimized' mesh and a mesh suitable for the two-layer approach.

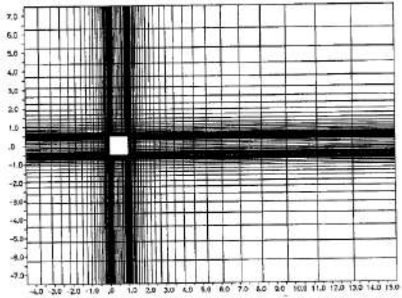


Figure 3. Computational mesh.

- Reduced disturbance by the cylinder presence

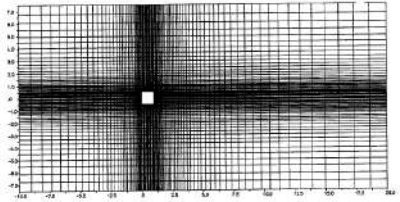


Figure 4. Optimized computational mesh.

- The results obtained with the optimised mesh have been compared with results obtained with meshes having 109x85 and 143x108 cells and no change was observed

Title: 2D LES of vortex shedding from a square cylinder
Author: D. Bouris, G. Bergeles —— **Year:** 1999 —— **Re** = 22000

The SIMPLE algorithm was employed to the staggered layout of variables on the grid and the third-order accurate, but unbounded, QUICK upwind scheme was used for the convection terms. Time discretization was First-order accurate fully implicit according to the Euler scheme.

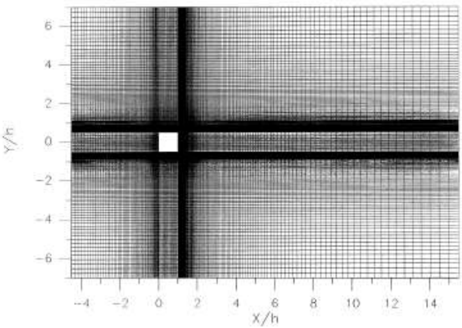


Fig. 1. Cartesian computational grid (300 × 350) used in the present study for large eddy simulation of the turbulent flow past a square rod.

Title: Computation of turbulent vortex shedding (LES)
Author: B. A. Younis and V. P. Przulj —— **Year:** 2006 —— **Re** = Several

Equations were integrated, term-by-term, over irregular cells formed from non-orthogonal meshes. Gauss's divergence theorem was used to relate the volume integrals to surface integrals

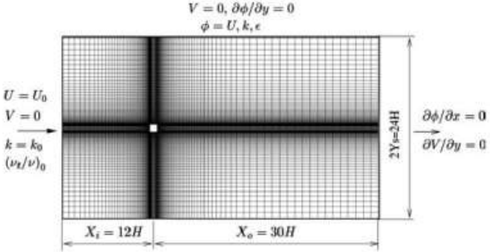


Fig. 2 Square cylinder. Grid $D1$ and domain boundaries

Table 2 Square cylinders. Parameters of numerical grids used

Grid [NI×NJ]	X_i/H	X_o/H	Y_s/H	f_{ex}	f_{ey}	$\Delta n_c/H$	$\%B_f$
$D0-106\times108$	12	30	12	1.10	1.10	0.02	4.17
$D1-139\times122(a)$	12	30	12	1.075	1.075	0.014	4.17
$D1-139\times122(b)$	12	30	12	1.0635	1.062	0.02	4.17
$D2-114\times92$	5.5	29.5	6	1.076	1.076	0.02	8.33

The blockage ratio (ratio of cylinder width to domain width at inlet) produced by using the above solution domain was $B_f = 4:17\%$. This is approximately equal to the values obtained in the experiments of Lee [4] and Bearman and Obasaju [5] but is smaller than that in the experiments of Lyn [6].

Title: Three-Dimensional Turbulent Vortex Shedding From a Surface-Mounted Square Cylinder: Predictions With Large-Eddy Simulations and URANS
Author: B. A. Younis and A. Abrishamchi —— **Year:** 2014 —— **Re** = Several

In contrast, in the URANS approach, the computational mesh size does not enter into the formulation of the turbulence model and, hence, successive mesh refinement will eventually yield results that are grid independent and, thus, largely free of numerical discretization errors. The computational resources needed to attain grid-independent solutions are significantly less than with LES.

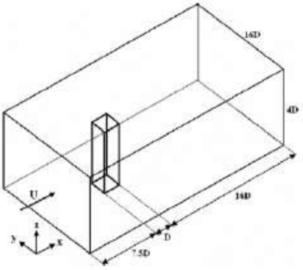


Fig. 1 Geometry, computational domain, and dimensions

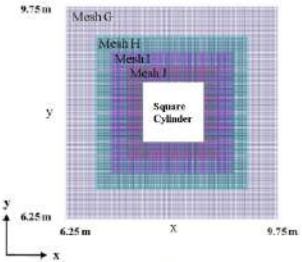


Fig. 2 Close-up of mesh in x-y plane

Title: Large-eddy simulations of flow past a square cylinder using structured and unstructured grids

Author: Yong Cao , Tetsuro Tamura —— **Year:** 2016 —— **Re** $= 22000$

The finite differencing method code with 4th order central scheme for the convective term is used for structured LES, while the open-source finite volume method code (OpenFOAM 2.3.0) with 1st–2nd order schemes is applied for unstructured LES.

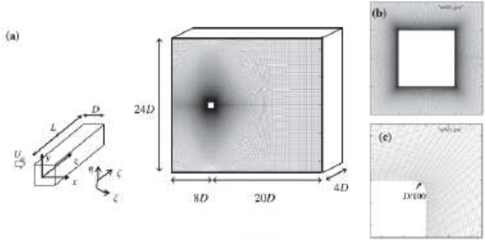


Fig. 1. Grid system on the generalized curvilinear coordinate used for the first group in Table 1: (a) whole grid system; (b) close-up around the cylinder; and (c) close-up around the corner with the radius of curvature of D/100.

Table 1
Study of grid-resolution sensitivity for structured LES.

Case	Grid	Δy	y^+	Δ_z	C_D	C_L'	St	$-C_{pb}$
1	$200 \times 200 \times 49$	5.6×10^{-4}	≤ 1.49	0.08	2.24	1.27	0.131	1.60
2	$300 \times 300 \times 81$	5.6×10^{-4}	≤ 1.49	0.05	2.21	1.26	0.132	1.59
3	$400 \times 400 \times 81$	2.8×10^{-4}	≤ 0.85	0.05	2.19	1.31	0.137	1.55
4	$300 \times 300 \times 129$	5.6×10^{-4}	≤ 1.49	0.03	2.22	1.30	0.140	1.59
DNS	$1272 \times 1174 \times 216$	$1.44 \sim 1.89 \times 10^{-3}$	≤ 0.56	0.0145	2.18	1.71	0.132	1.58
LES	–	–	–	–	2.02–2.77	1.15–1.79	0.09–0.15	–

Firstly, “CR”represents the mesh region near cylinder is refined, provided that the grid topology remains structured.

By contrast, “WR”and “CWR”use the hybrid mesh in order to take advantage of the flexibility of unstructured grids to refine where we want. Specifically “WR”represents the wake region is r efined, provided that the total cell number remains similar to the first group.

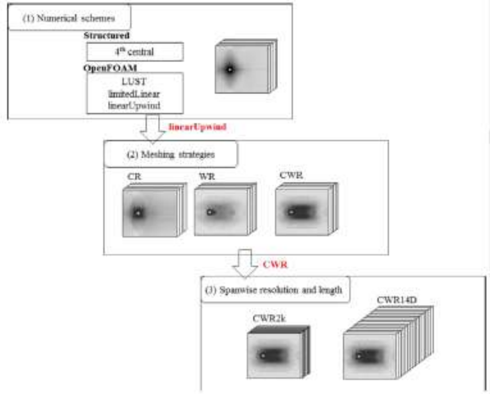


Fig. 3. Schematic diagram for interpretation of relationship between computational cases.

“CWR”means both the cylinder and wake are refined resulting in the similar total cell number as “CR”.

This study really goes in depth into the meshing of the square cylinder problem. The meshes are gradually reduced to study the grid dependence behaviour.

Title: Unsteady RANS simulations of flow around rectangular cylinders with different aspect ratios

Author: Xinliang Tian a,n, Muk ChenOng b, Jianmin Yang a, Dag Myrhaug —— Year: 2016 —— **Re** = 21400

Model: SST

The drag forces acting on the cylinders with high aspect ratios (R = 1, 0.8 and 0.6) are well predicted by the simulations; however, the drag forces are overpredicted for low aspect ratios (R=0.4, 0.2, 0.1, 0.05).

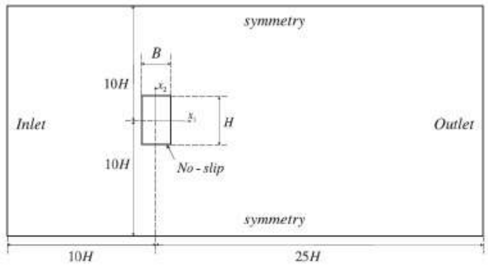
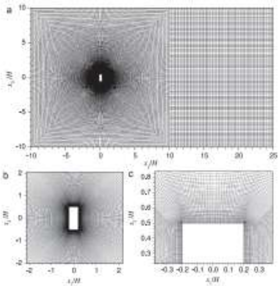


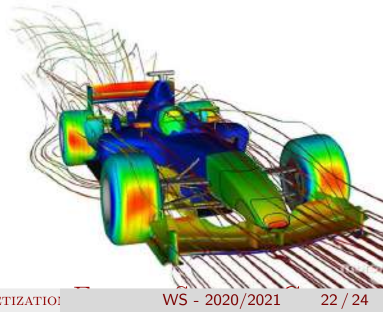
Fig. 1. Computational domain and boundary conditions.



Conclusions

- The selection of the computational domain is a decision that could severely affect the results.
- The cylinder should not be affected by the boundaries and the inflow should not be perturbed by its presence.

- Modern studies implement the 3D analysis of the problem and most of them use improved meshing strategies.
- Modern approaches also tend to implement FEM but FVM is mostly predominant.



Questions?

Bibliography

[1] S. V.Patankar, Numerical Heat Transfer and Fluid Flow, 1st ed., 1980

[2] Joel H. Ferziger, Computational Methods for Fluid Dynamics, 4th ed., 2020

[3] CTTC,Course on Numerical Methods in Heat Transfer and Fluid Dynamics. Fractional StepMethod. ESEIAAT

[4] Lee BE (1975) The effect of turbulence on the surface pressure field of square prism. J Fluid Mech 69:263–282

[5] Bearman PW, Obasaju ED (1989) Transverse forces on a circular cylinder oscillating in-line with a steady current. In: Proceedings of the 8th international conference on offshore mechanics and arctic engineering, Hague, March 19–23, pp 253–258

[6] Lyn DA (1992) Ensemble-averaged measurements in the turbulent near wake of a square cylinder: a guide to the data, Report CE–HSE–92–6, School of Civil Engineering, Purdue University, USA

F1 Image Conclusions courtsey of Sauber Petronas.