LANDMARK UNIVERSITY, OMU-ARAN

LECTURE NOTE 2a
COLLEGE: COLLEGE OF SCIENCE AND ENGINEERING
DEPARTMENT: MECHANICAL ENGINEERING
Course code: MCE 521
Course title: ADVANCED COMPUTATIONAL DYNAMICS.
Credit unit: *2 UNITS.*
Course status: *compulsory*
ENGR. ALIYU, S.J

BACKGROUND FOR CFD.

Computational Fluid Dynamics (CFD) is the science about calculation of fluid flow and related variables using a computer. Usually the fluid body is divided into cells or elements forming a grid. Then equations for unknown variables are solved for each cell. This requires a substantial amount of computing resources. Therefore, this science has not progressed to a practically applicable stage until recently. In coming years, CFD will increasingly be used in hydraulic and sedimentation engineering. It is therefore important that engineering students are given insight into this topic.

Physical models are costly and time-consuming, compared with CFD models. A particular problems with physical model studies

[redacted] has been the simulation of suspended sediments. Cohesive forces are present for fine particles necessary when scaling down sediment sizes for simulation of sand traps. Calculation of trap efficiency of sand traps is one of the first areas where numerical models were used instead of physical models. Since then the CFD models have expanded to include sediment transport with time-dependent moveable bed, trap efficiency, deposition, erosion, local scour, turbidity currents, flood waves, spillways and head loss in channels and tunnels.

GRIDS.

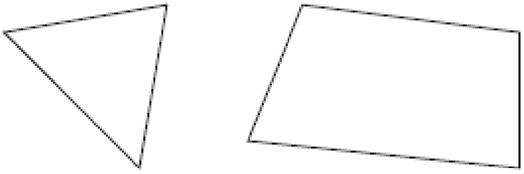
A basic concept of CFD is to divide the fluid geometry into elements or cells, and then solve an equation for each cell. In the following text, the word cell will be used instead of element, to avoid confusion with the finite element method. The algorithms described in the following chapters are based on the finite volume method. [redacted]

CLASSIFICATIONS

Grids can be classified according to several characteristics:

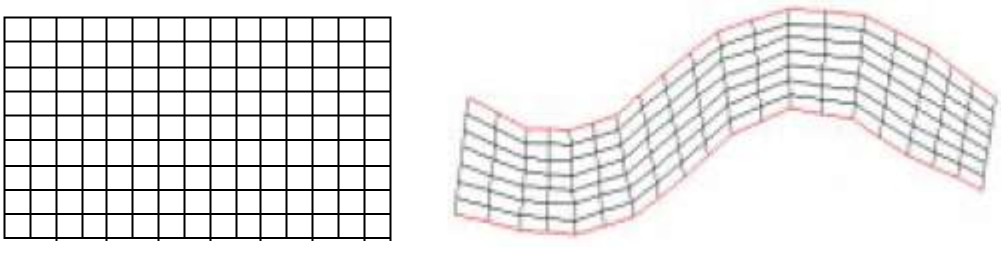
- shape
- orthogonality
- structure
- blocks
- position of variable
- grid movements

The **shape** of the cells is usually triangular or quadrilateral:



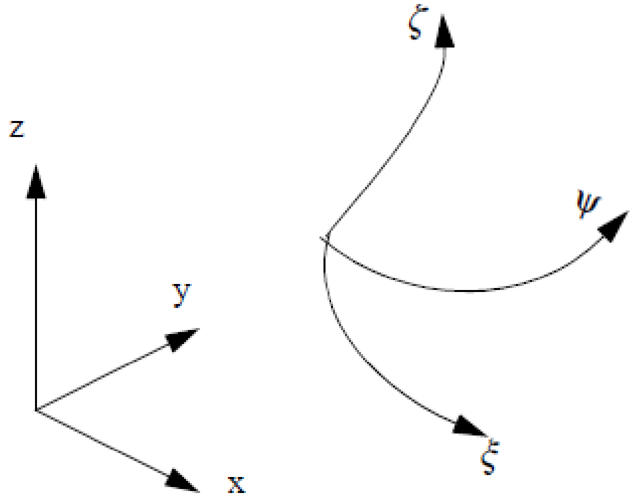
Triangular and quadrilateral shapes

The **orthogonality** of the grid is determined by the angle between crossing grid lines. If the angle is 90 degrees, the grid is orthogonal. If it is different from 90 degrees, the grid is non-orthogonal.



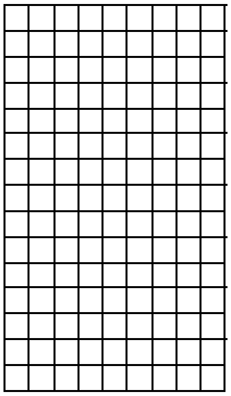
Orthogonal grid **Non - Orthogonal grid**

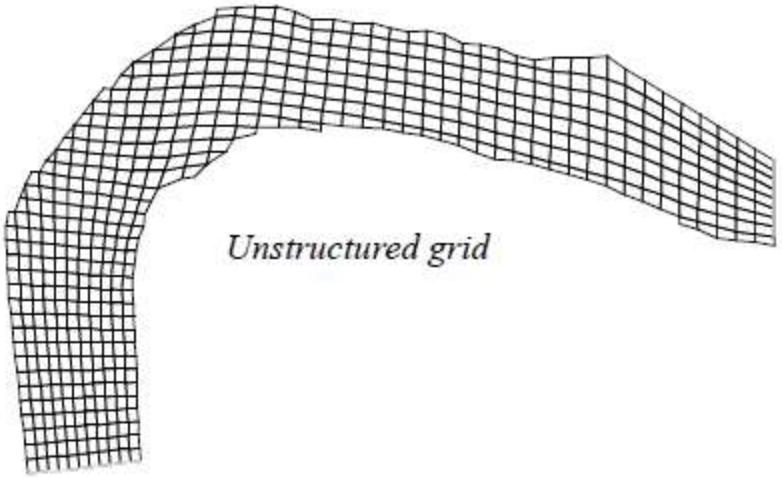
For non-orthogonal grids, a non-orthogonal coordinate system is often used to derive terms in the equations. The coordinates then follow the grid lines of a structured grid. The three non-orthogonal coordinate lines are often called ξ,ψ,ζ, corresponding to x,y and z in the orthogonal coordinate system. This is also shown on the figure below:



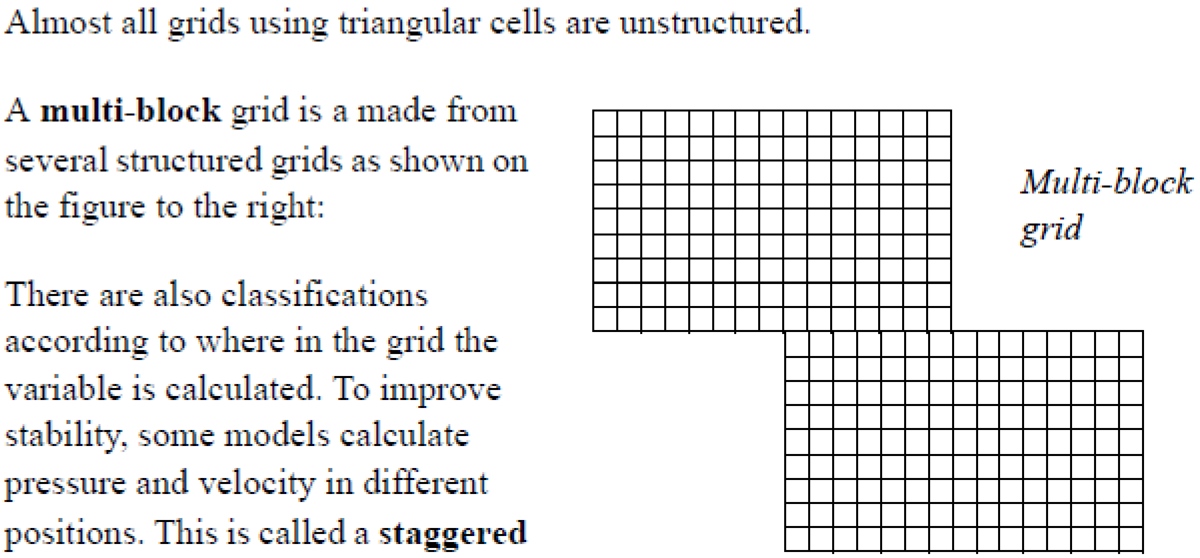
Non-orthogonal coordinate system following the grid lines.

Grids can be **structured** or non-structured. Often a structured grid is used in finite volume methods and an unstructured grid is used in finite element methods. However, this is not always the case. The figure below shows a structured and an unstructured grid. In a structured grid it is possible to make a two-dimensional array indexing the grid cells. If this is not possible, the grid is unstructured.





Structured grid

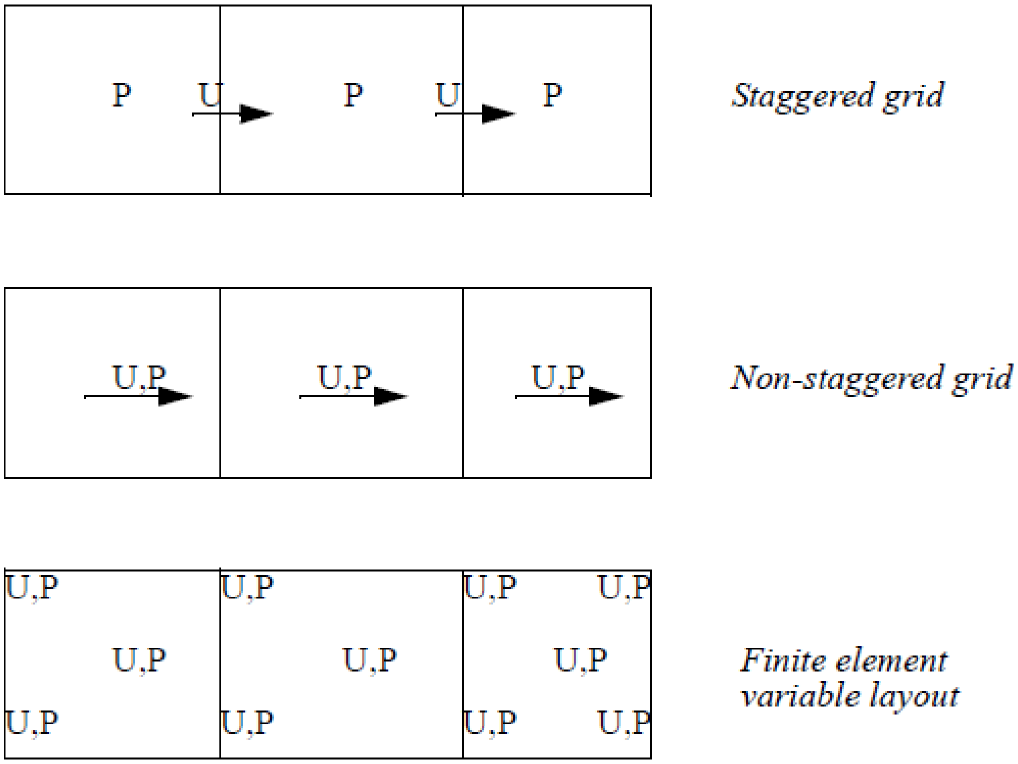


Almost all grids using triangular cells are unstructured.

A **multi-block** grid is a made from several structured grids as shown on the figure to the right:

Multi-block grid

There are also classifications according to where in the grid the variable is calculated. To improve stability, some models calculate pressure and velocity in different positions. This is called a **staggered** grid. A **non-staggered** grid is used when all variables are calculated in the same location, most often the centre of each cell. These terms are most often used in the control volume terminology. In finite element methods, the variables are most often calculated at grid intersections, and sometimes also in the centre of a cell. The following figure shows this with circles indicating the locations of the calculation points. P denotes a point where pressure is calculated, and U denotes a point where velocity is calculated.



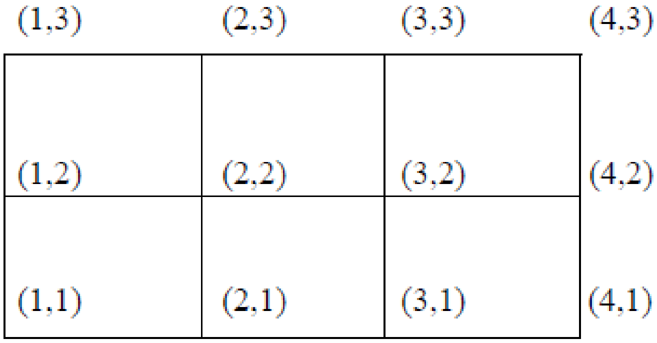
An **adaptive** grid moves according to the calculated flow field or the physics of the problem. When the water surface or the bed moves during a time step, it is possible to make the grid move accordingly, to calculate the situation for the new geometry. Thereby time-dependent calculations of bed changes and water levels can be done.

An adaptive grid is used to model bed changes in for example sediment deposition, reservoir flushing or local scour. It is also used to model changes in the water surface when for example calculating a flood wave.

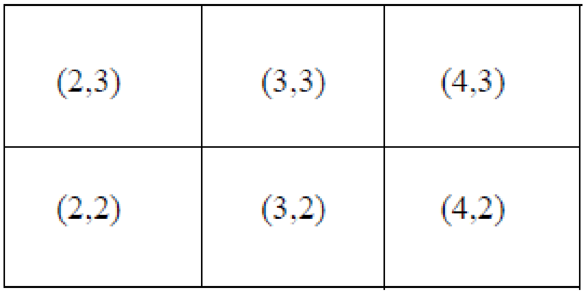
STRUCTURED GRID NUMBERING

There are several ways to identify cell and grid lines in a structured grid. When using some CFD programs, it is important to know the numbering system for each particular program in order to identify the regions of inflow/outflow and outblocking procedures. The numbering system described in the following is used by the SSIIM program.

In a structured grid, there will always be one more grid line than grid cells in any given direction. Because boundary conditions are also needed, there will be a grid cell with zero thickness at the walls. The number of this cell is 1. The number of the first cell inside the grid is therefore 2. The figure below shows a two-dimensional view of the grid, with numbers.



Grid line numbering



Grid cell numbering

The variable in centre of the “first” grid cell has number (2,2). The variables on the boundary are only used as boundary conditions. The variables in the corners: (1,1), (1,5), (5,1) and (5,5) are not used for anything.

Note that different computer programs may have different grid numbering systems.

GRID QUALITIES.

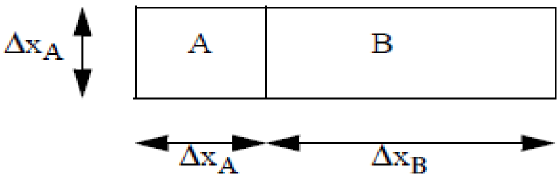
The accuracy and convergence of a finite volume calculation depends on the quality of the grid. Three grid characteristics are important:

- non-orthogonality
- aspect ratio
- expansion ratio

The non-orthogonality of the grid line intersections is the deviation from 90 degrees. If the grid line intersection is below 45 degrees or over 135 degrees, the grid is said to be very non-orthogonal. This is a situation one should avoid. Low non-orthogonality of the grid leads to more rapid convergence, and in some cases better accuracy.

The aspect ratio and expansion ratio is described in the figure below:

The figure shows two grid cells, *A* and *B*. The length of the cells are Δx_A and Δx_B.



Expansion/aspect ratio

The **expansion ratio** of the grid at these cells is $\Delta x_A/\Delta x_B$.

The **aspect ratio** of the grid at cell A is $\Delta x_A/\Delta y_A$.

The expansion ratio and the aspect ratio of a grid should not be too great, in order to avoid convergence problems and inaccuracies. Aspect ratios of 2-3 should not be a problem if the flow direction is parallel to the longest side of the cell. Experience shows that aspect ratios of 10-50 will give extremely slow convergence for water flow calculations. Expansion ratios under 1.2 will not pose problems for the solution. Experience also shows that expansion ratios of around 10 can give very unphysical results for the water flow calculation.

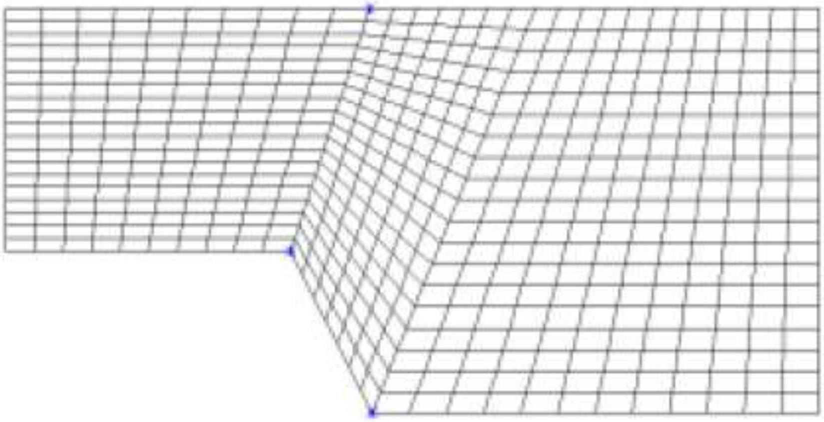
ELLIPTIC GRID GENERATION.

Elliptic grid generation is a method to distribute internal points smoothly in a structured quadrilateral grid. This is done by solving a Laplace or Poisson equation for the location of the grid line intersections:

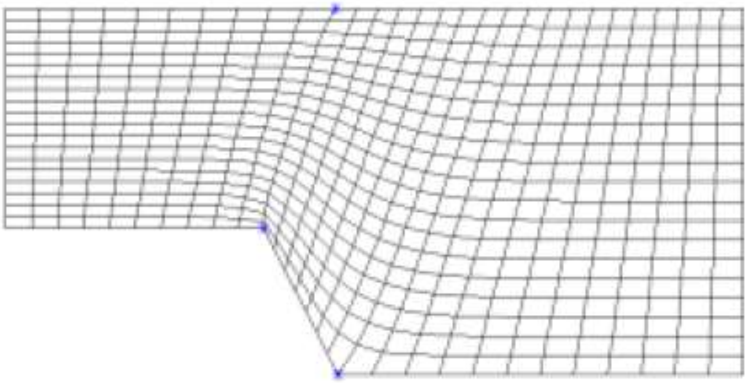
$$\nabla^2\xi^i = P^i \qquad \ldots\ldots\ldots\ldots \quad \mathbf{1}$$

The location of the grid lines are denoted ξ. P is a source term used for attracting grid lines to a side or a point.

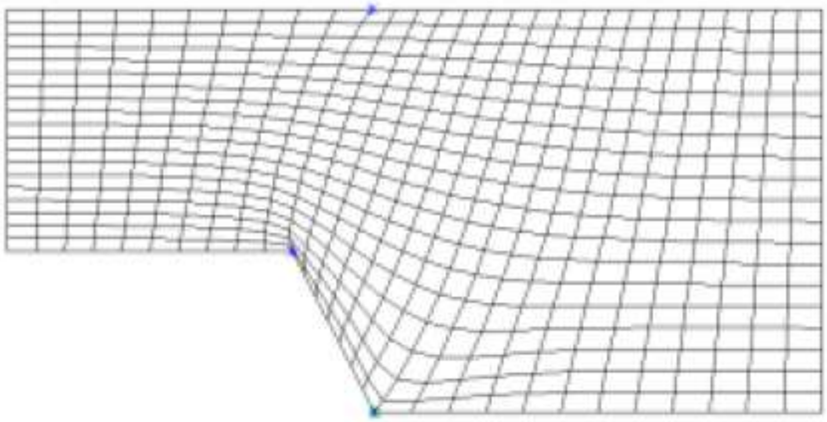
An example of using elliptic grid generation is given below:



This grid is made by use of transfinite interpolation. This means that straight lines are made between the points on the wall.



Elliptic procedures are used to obtain the grid shown above, with no attractions. This means P=0 for Equation [redacted]1.



Looking at the lower point of the step, the cells are smaller closer to the wall. This is due to attraction functions. P ≠ 0 for Equation [redacted]1.

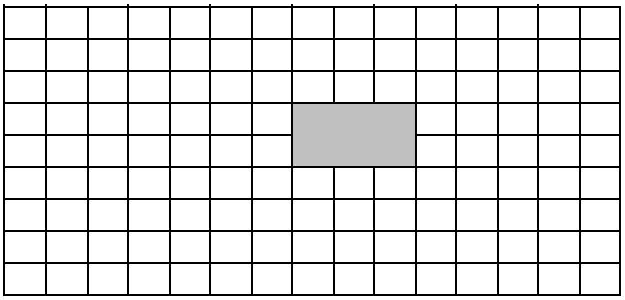
PRACTICAL ADVICE FOR GENERATING STRUCTURED NON-ORTHOGONAL GRIDS.

Making a good grid is a considerable part of the work for CFD calculations for complex geometries. Experience seems to be very important for this task, so the first advice is to practice making grids.

Before starting to make the grid, some information is necessary:

1. How many grid cells are available for the grid? This can be calculated from the capacity of the computer and the expected computational time.
2. Drawings of the geometry are important, where boundaries and the inflow/outflow regions are shown together with possible islands or other **outblocked** regions.

An **outblocked** region is a part of the grid where water is not allowed to flow. It can be used for making islands or obstacles in the flow. An example is given below with flow around a square obstacle:



Grid with an outblocked region of 3x2 cells

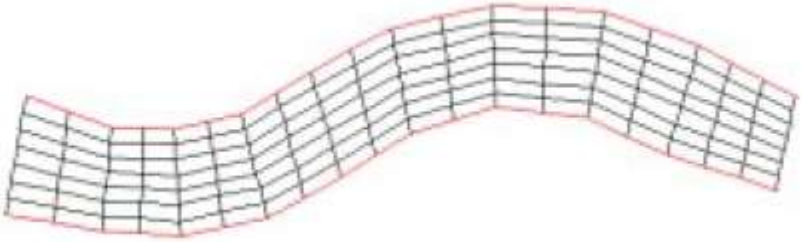
Note that not all computer programs have the option of blocking out parts of the grid.

If the geometry is complex, one should make a paper sketch of the grid before starting to use the computer. The paper sketch should be reasonably large in order for the details to be visible. The boundary of the grid should be marked with one colour, and inflow/outflow regions with another colour.

One should start with deciding the general structure of the grid: where the four sides of the grid should be located. If outblocking is needed and possible, this should be decided together with the location of the outblocking in the grid. Then one should start drawing the grid, using a pencil and an eraser, as modifications are always needed.

One should preferably start making a very coarse grid, maybe with only 6-12 cells, and making the grid finer afterwards. When drawing the grid, the following points should be kept in mind:

1. The grid lines should follow the stream lines as much as possible
2. The grid line intersections should be as orthogonal as possible
3. The grid aspect ratio should not be too great
4. The grid expansion ratio should not be too great
5. There should be higher grid densities in areas with high velocity/concentration gradients. In a not too complex part of a river, the gradients in cross-streamwise direction are larger than in the streamwise direction. The grid cells can then be longer in the streamwise direction than in the cross-streamwise direction. An example is given in the figure below, where the aspect ratio for the cells are above unity.



After having made the paper sketch of the grid, the tools of the CFD computer program can be used to make the final grid. Note that the same points as given above apply when using the grid tools of the CFD program.