INTRODUCTION TO CFD IN NAVAL HYDRODINAMIC: A little review

José Enrique Gutiérrez Romero

January 17, 2011

Revised by: Julio García Espinosa & Blas Zamora Parra

1.- ¿What are CFD?

The CFD acronym means "Computational Fluid Dynamics". This is a very useful tool to solve the basics equations which model the flow movement. The majority of these equations do not have any analytical solution for these reasons we resort to numerical analysis with CFD. The objective of these tools is to solve approximate (numerically) the flow basic equations whose solutions give us the movements and other characteristics of the flow.These techniques discretise the spatial and time domain for reaching the solution.

2.- Historical blackground.

The tools development of CFD started in the 1950s with the development of computers. The Finite Element Method (FEM) Finite Difference Methods (FDM) are the basic techniques to solve the equations raised by the mathematics and physics. Firstly the Finite Difference Methods have dominated the calculus because of the Finite Element Methods were more computationally expensive than FDM and they were more difficult to formulate than FDM. Nowadays this situation has changed, today new methods and formulations with FEM have appeared and this situation has caused the obsolescence of FDM. Thus, other methods have got popularity, for example finite volume methods. This constituted a natural extension of FDM into non-structured meshes. FVM are temporally situated before FEM into techniques CFD . These types of methods permit to integrate the mass and momentum equations in each cell of mesh and they permit to approximate their value in the center of the cell. Other new methods we find Smooth Particles Hydrodynamics (SPH). This method establishes a new field in existing techniques.

The problem of ship resistance, understanding resistance how the flow effect over the surface of the body which is submerged in the flow was one of the pillars of genesis of naval hydrodynamics. The interest in this new problem was caused by its applications in several fields such as hydrodynamics, windmills and other devices which were submitted to environmental forces.

The first approach to explain the body drag in the XVII century followed different ways until the occurrence of the basic equations of the flow movements, although the lack of calculus capacity kept the improving on this way.

Late of XVII century the “impact Theory” carried out. This theory considered two hypotheses. On the one hand the theory supposed that the fluid was constituted by bunch of individual and independent particles and on the other hand considered that summation of all the impact in the body was equal of the body drag.

This theory did not explain the several resistance aspects. Later in the middle of XVIII century Euler established a new technique which was approximate to current theories. In the same century Jean le Ron d´Alambert exposed a new idea about the flow movement around the submerged body. In this theory the fluid is supposed to be animated with uniform movement. Then when the flow starts to go up the body, the flow varies its way around the body. Finally the stream lines of the body will join together upstream.The flow effect in the body will be made by the pressure rather than the fluid impact. However two premises will be necessary to obtain the fluid field: the continuity and dynamical. The development of this field was paralyzed until the beginning of computer.

3.- ¿Where are CFD used?

Nowadays these techniques have been in almost fields ranging from medical research until engineering. In hydrodynamics several important aspects are taken into account:

- Resistance and propulsion.
- Maneuverability
- Seakeeping
- Propeller design
- Others applications: piping...

Resistance and propulsion.

The basics applications of these techniques are centered in this field. Here the engineers use BEM calculus where the viscous effect and the wake generation in the free surface are despised. This provides a simple, fast and approximate calculus. At the end of the 90 years all these simplifications were dismissed because of the increasing of the potential calculus in computers. Nowadays there is a great variety of codes which permits viscous calculation, this permits to reach best results that are in concordance with real experiment.

Maneuverability

In this field, today the new IMO regulations get more and more importance. For this reason through CFD analysis we can obtain the different forces actuating over ruder surface. This permits to calculate the different moments in the ship. Thus we will evaluate the maneuverability of the ship.

Seakeeping

Here the CFD codes constituted unripe part in the current techniques although today new codes appear with new features. For several years there have been commercial codes into BEM field, one of theses is WASIM. These new codes includes permit to calculate the ship movement in different seas. However the problem of ship Seakeeping presents a great difficulty because it is necessary to provide with all the features of sea and a lot of Knowledge about this field will be necessary.. We will be need several input data such as height of waves, period, and scattered waves, mass added, ship characteristics...The future in this field is offshores structures. Today the wind energy generation probably constitutes the principal battle field of Seakeeping.

Propeller design

The usual techniques in this field are viscous CFD techniques because these permit to improve the results and permit to approximate to reality. Here employed the basics techniques are Boundary Element Methods. These usually utilize the supportive profile theory to calculate the different parameters of propeller.

To finish with this part we will probably encounter CFD techniques in other field, for example:

- Piping
- Air flow
- Fire protection of ship...

4.- CFD advantages versus disadvantages

The *advantages* of CFD techniques will probably be summarized in the following lines:

- A great time and cost reduction in the new designs
- There is a possibility to analyze different problem whose experiments are very difficult and dangerous.
- The CFD techniques offer the capacity of studying system under conditions over its limits. These conditions will probably be dangerous to experimental.
- The level of detail is practically unlimited. The experimental methods are more expensive with the increasing of measure point. However in CFD techniques permits to generate several pieces pf information without any cost. This offers the possibility to carry along a lot of parametric studies.
- The product gets added value. The possibility to generate different graphs permits to understand the features of result. This encourage to buy a new product.

On the other hand, all on these tools are not advantages. The principal *disadvantage* of CFD tools is the great knowledge about the basic equations that describe the fluid movement and certain physical phenomenon necessary to solve a new problem.

- Other disadvantage is the accuracy in a result. In several situations probably we will not obtain a successfully result. In these cases the mistakes will be important. Even though if the user is untrained.
- It is necessary simplify to mathematically the phenomenon to facilitate the calculus. If the simplification has been good the result will be more accurate.
- Today there are different incomplete models to describe the turbulence, multiphase phenomenon and other difficult problems.
- Finally, it is important to mention that the untrained user of CFD codes has the tendency to believe the output of PC is always true.

5.- Approximation to reality

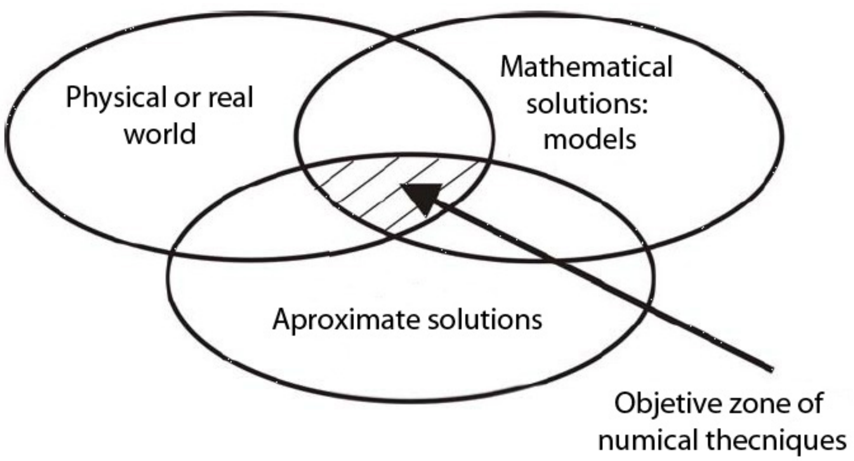
The idea about this concept results very clearly. They are the different approximations made by engineering to solve the fluid equations.

Once known the different equations of the physic phenomenon to study it will be necessary to introduce all the possible simplifications in a new problem. This helps us obtain the best result with the less cost and difficulty. So, we don't understand the analytical solutions of the problem under study previously for these do not reasons the simplifications are very difficult to introduce in the new problem.

It will also be necessary to dispose the equations of the physical phenomenon to make the model about the problem. Most of these equations are differential equations with nonlinear terms and other are partial equations. All of these equations will keep off to obtain an easy solution of the problem. Other important aspect to obtain an accurate solution of the problem is the spatial discretisation of the same. The fist step will always be the problem discretisation. Firstly the geometry will be divided in a lot of little parts, this permits to solve the problem easily. Although the fluid field is continuous, the equations are not be solved in all the fluid point, so, for this we need to discretise the entire domain in several points where the equations will be solved. This step is named as ***spatial discretization***.

In other way when the phenomenon is constituted by time dependent equation neither it is possible to study the continuous phenomenon. In this situation the engineer establishes a ***temporal discretization***.

Finally in this part in government equations all of the unnecessary terms will be eliminated . So, we obtain a new level in approximation in a phenomenon called ***dynamical discretization***.



6.- Classifications of CFD tools

Into the naval field, the CFD tools can be classified in two different groups: those with potential flow, to say the tools which do not consider the fluid viscosity. We also have two more types: with or without free surface into these types. On the other hand we encounter other tools which consider the fluid viscosity. Here also we find the same types: with or without free surface.

7.- Validation of CFD code. The quality importance of CFD´s

Today the quality concept reaches the computer systems and particularly the computer programs. After reached and promoted the quality concept into "grey material" in the hardware through ISO rules, for this it is necessary to face the problem of CFD validations, mainly in those tools which are designed to hull and propeller optimization.

The CFD validations could be understood as a demonstration which permits to verify the result obtained in a computer. This result should be similar to experimental results. In this way the best orientation to show the goodnesses of CFD tools is an experiment.

- **Functionality**: The CFD programs must provide the information using the same parameters and nomenclature which are compatible with the current techniques employed in industry.

- **Reliability:** These types of codes have to calculate the result into a confidence range and they should reach an acceptable precision level.

- **Usability:** This feature evaluates the capacity to use the program, to say the facility of user to understand the capacities of program and the logical of his code in the inputs and outputs.

- **Efficiency**: The efficiency of the program should be measured not only in time wasted for the CPU and GPU but also in all the human resources employed to obtain a good result.

- **Maintainability:** This part tries to show the short life of these codes and the necessity to update these. These updates permit to be suitable in current techniques.

- **Portability:** A CFD program should be installed in different operating system and give us the possibility to transfer.

8.- Some physics and mathematical models employed in CFD analysis.

First of all we can say that the momentum equation and mass conservation equations are more complicated than it seems. The experience has demonstrated that the Navier – Stokes equation describes the movement of fluid accurately, however today only a few problems have been solved in simple geometries. This only has a few applications in the industry.

8.1.- Potential flow model

These models describe the behavior of an ideal problem. In this case the basic theory to calculate the features of fluid starts with the velocity potential and it establishes an added simplification to permit to calculate steady flows. So into these models we find different developed methods used by different authors. Michel in 1898 obtained some formulation to calculate the wave making. On the other hand Haverlock in 1932 also gave a solution to get the wave resistance. Other authors such as Hess & Smith provided us with method to obtain the potential velocity. Theses methods establish the following problem:

$$\nabla^2\phi = 0 \qquad (Laplace\ equation)$$

To solve this problem it will be necessary to apply a boundary condition in a free surface of fluid and in ship hull.

Boundary condition one:

$$\begin{cases} \nabla^2\phi = 0 \\ \phi = U_x + \phi_p \end{cases} \Rightarrow \nabla^2\phi_p = 0$$

In a ship hull we applied the following boundary condition:

$$V_n = \frac{\partial\phi}{\partial n} |_s = 0 \qquad \frac{\partial\phi}{\partial n} |_s = -U\frac{\partial x}{\partial n} |_s$$

Kinematic boundary condition:

$$\left(U + \frac{\partial\phi_p}{\partial x}\right)\frac{\partial h}{\partial x} + \frac{\partial\phi_p}{\partial y}\cdot\frac{\partial h}{\partial y} + \frac{\partial\phi_p}{\partial z}(-1) = 0$$

Dynamical boundary condition:

$$p+\frac{\rho}{2}\left[\left(U+\frac{\partial\phi_p}{\partial x}\right)^2+\left(\frac{\partial\phi_p}{\partial y}\right)^2+\left(\frac{\partial\phi_p}{\partial z}\right)^2\right]+\rho gh=p_{atm}+\frac{1}{2}\rho U^2$$

8.2.- Ideal flow model

When the Reynold number is sufficiently high in some problems of fluids mechanics it is easy to despise the viscous effects and other conduction effect to facilitate the resolution of equations. Thus we can solve more easily the equations because we eliminated second order diffusive terms in the differential equation. This permits to obtain first order equations which facilitate the resolution of the problem.

With the last hypothesis, we obtain the equations of ideal flow which are called Euler equations. The models of Euler equations take the following expressions:

Continuity:

$$\frac{d\rho}{dt}+\rho\nabla\cdot\overrightarrow{u}=0$$

Momentum:

$$\rho\frac{\partial\overrightarrow{u}}{\partial t}+\rho\cdot\left(\overrightarrow{u}\cdot\nabla\right)\overrightarrow{u}=-\nabla p+\rho f_e$$

8.3.- Rotational, steady and ideal flow models

These types are very similar to ideal models. Here the engineering tries to reduce the number of variables which are involved in adding the vorticity of momentum and energy equations. Here we do not consider either the waste by viscosity in the boundary layer or turbulence effects.

8.4.- Direct solution of Navier – Stokes equations

The Navier- Stokes equations constituted the correct flow modeling of Newtonian fluid, the viscous and term effect are included in the same equations. If the equations are perfectly solved the effect of turbulence and boundary layer are added in the solution. However to obtain a correct solution the smallest spatial and temporal discretization will be necessary. This means that is impossible in the current industry. Today this is possible in the supercomputers and the solution will probably be obtained in the future in a normal computer.

Now the numerical solution is possible today if suitable models are used to solve the equations and simulate the turbulence effect and boundary layer.

8.5.- Parabolic models of Navier – Stokes equations

The parabolic models are developed to model and calculate supersonic and hypersonic flows where the capture of shock waves, pressure gradient, superficial viscous efforts and heat transfer are the most important objectives in industry.

The parabolic government equations are obtained using the Navier – Stokes equations and considering the following features:

1. Steady flow
2. The viscous efforts gradient are despised in the stream lines direction.
3. The pressure gradient are approximated with their values in the nearest boundary layers.

8.6.- Incompressible flow models

One fluid is incompressible when it s density is constant and the pressure changes and time independent. When the fluid is isotherm, the government equations are notable simplified and the solution of several variables becomes temperature independent. The equation system required is reduced to continuity equation and momentum equation which are expressed with dimensionless term. Theses adopt the following expressions:

Continuity:

$$\nabla \cdot \overrightarrow{u} = 0$$

Momentum:

$$\frac{d\overrightarrow{u}}{dt} = -\frac{\nabla p}{\rho} + \frac{1}{Re}\nabla^2\overrightarrow{u} + \frac{1}{Fr}\overrightarrow{g}$$

In this way; the continuity and momentum equation are energy independent, for these reasons there is not any necessity to solve the equations to obtain the velocity and pressure fields. However in spite of this advance, the equations become too “rigid” and in fact the solutions of these are more complicated than others.

8.7.- Models to solve the turbulence

The Reynold number gives us the importance of inner forces associated with convective effects and viscosity forces. We know before a less Reynold number the critic flow is steady and the adjacent flow layers slide on each other in order. In this case the flow is called ***laminar flow***.

If the flow reaches and passes a critical Reynold number, several disturbances will be show which will cause a radical change of the behavior of the flow. The movement will become unsteady even if it has a steady boundary conditions. This flow is called ***turbulence flow***.

The state of flow movement when the flow condition and its properties (for example: pressure, velocity...) fluctuate untidily is defined turbulence. This is a unsteady state from a macroscopic point of view. The values of the flow properties vary untidily and randomly and they depend of the velocity, pressure, time position...

The movement description of the flow particles is highly complex because of turbulence. It also constitutes a problem without solution from the point of view of numerical methods. So, the turbulence simulation is one key of CFD simulation:

Direct simulation (DS).

This method does not use any turbulence method, however it uses temporal and spatial discretisations which are capable of simulating a concrete problem. Today the study of the direct simulation of Navier-Stokes equations is accessible for limited cases. The first solution of this type was achieved in Stanford University in 1981.

Large Eddy Simulation (LES).

These types of numerical techniques reduce the complex of the government equations. These only consider one of part of turbulence flow. The energy exchange into the "large-scale fluctuation" and the effect of little eddies of turbulence are studied into these . These types of models are situated between Direct Simulation and Reynold Averaged Navier Equations. The last type extends the temporal average into specific and basic turbulence effects. Although these types of turbulence models are used in simple geometries because they require a high level of calculus capacity.

Unsteady Reynolds Averaged Navier Stokes (URANS).

These types of model are similar to RANS methods, however these types of methods work with transitory scheme instead of RANS methods. Thus the solution of the problem depends on the time. Today URANS methods are suffering a great development.

Reynolds Averaged Navier Stokes (RANS).

The RANS methods have been widely studied and they are very useful to solve the most of practical problem solved with numerical methods.

The procedure of average the movement law of the flow particles is introduced to obtain the averaged and random behavior of the several variables. The starting point into these methods is very easy. The procedure tries to split the variables into average values and random values. So, there are three types of RANS: time averaging, spatial averaging and ensemble averaging.

Time averaging is appropriated for stationary turbulence, i.e. a turbulence flow that, on the average, does not vary with the time. For such a flow, we express an instantaneous flow variable as *f(x,t)* . Its time average, $F_T(x)$, is defined by:

$$F_T(x) = lim_{T\to\infty} \frac{1}{T} \int_t^{t+T} f(x,t)dt$$

Spatial averaging is appropriated in homogeneous turbulence, which is a turbulence flow that, on the average, is uniform in all directions. We average over all spatial coordinates by doing a integral volume. The spatial averaging, $F_V(x)$ is defined by:

$$F_V(x) = lim_{T\to\infty} \frac{1}{V} \iiint f(x,t)dV$$

Ensemble averaging is the most general type of averaging. As an idealized example, in terms of measurement from N identical experiments where $f(x,t) = f_n(x,t)$ in the n^{th}, the average F_{E} where:

$$F_E(x,t) = lim_{N\to\infty} \frac{1}{N} \sum_{n=1}^{N} f_n(x,t)$$

When the turbulence is homogeneous and steady, we may assume that these three average are equal. This assumption is known as ergodic hypothesis.

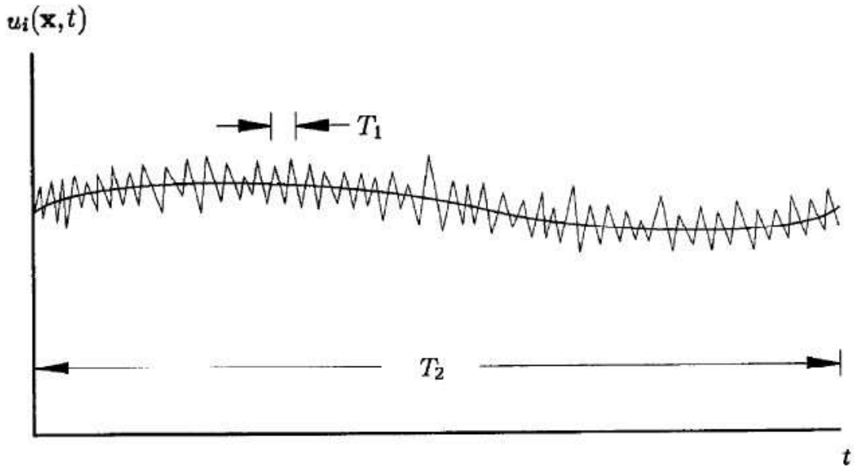
For example, the descomposition of velocity will be:

$$\overrightarrow{u} = \overline{\overrightarrow{u}} + \overrightarrow{u'}$$

Where middle component of velocity is obtained making the integral of instant velocity:

$$\overline{\overrightarrow{u}} = \frac{1}{T} \int_0^T \overrightarrow{u}(t)dt$$

Supposting, that the integral period is large enought compared with temporal turbulence scale, but little enought to capture any unsteady phenomenon different from turbulence. The utilization of these methods is suitable, so the most of unsteady phenomenon in Fluid Mechanics has been placed out of range of normal range of turbulence.



The process of time averaging of differential equations results into several terms denominated Reynold Stresses, which involve averages of the products of fluctuating velocity components. To obtain the relationship is to necessary introduce a new model, denominated turbulence models or closure model. The different practical possibilities of these models will be analiced in the following paragraphs.

So as, the Navier Stokes equations are averaged above scales of the turbulence fluctuations (RANS). These methods cause a field of averaged turbulence and simulated which is more uniform than real field, therefore, the number of points of the spatial and temporal discretization necessary to obtain the variables are reduced highly.

So a turbulence model is a numerical procedure to obtain the averaged values of variables of fluctuations. This permits solve the government equations more easily. If a turbulence model is exact, simple and economic, the turbulence model will be useful into a general CFD. The most common turbulence models are the following:

RANS-based turbulence models

Algebraic models.

1. Cebeci-Smith Model
2. Baldwin-Lomax Model
3. Johnson-King Model
4. Rought wall dependent Model
5. Mixing Length Model.

One equations models.

1. Prandt Model

2. Baldwin-Barth Model
3. Spalart-Allmaras Model

Two equations models.

1. k-epsilon Model
2. k-omega Model

Two equations models with restrictions and limits

v2-f Models (Non linear eddy viscosity models)

Reynolds Stress Models

Large eddy simulation (LES)

1. Smagorinsky-Lilly model
2. Dynamic subgrid-scale model
3. RNG-LES model
4. Wall-adapting local eddy-viscosity (WALE) model
5. Kinetic energy subgrid-scale models
6. Near-wall treatment for LES models

Others models

Detached eddy simulation (DES)

Direct numerical simulation (DNS)

Turbulence near-wall modeling

Turbulence free-stream boundary conditions. Then we will deepen in to the most common and known turbulence models.

Algebraic Models

The algebraic models of turbulence also called models of cero equations.These types of models do not need additional equations and they are directly solved over the fluid variables. They utilize a Boussinesq approximation to obtain the Reynold Stresses. In these models the eddy viscosity and mixing length depend on the fluid and to solve them will be necessary to specify these two variables. For this reasons theses models are not suitable in the most of applications because of they will consider convection and diffusion of turbulence energy. The algebraic models are very simplex , so the user of CFD should take care to utilize these models in complex geometries and process because the result probably will not be valid, for this reason they are not valid to use in general analysis with CFD.

Prandt Model

In 1925 Prandt proposed the hypothesis and bases to develope this new method. Prandtl formulated a turbulence model whereby the particles coalescence and joined a group of particles which moved as a singleness. He also visualized that the shear flow has a linear profile and the groups of molecules kept their momentum in the X direction to a distant in a Y direction. This direction was denominated how "mixing length" l_{mix}. So he postulated that:

$$\tau_{xy} = \frac{1}{2}\rho u_{mix} l_{mix} \frac{dU}{dy}$$

where mixing velocity take the following expression:

$$u_{mix} = Constant \cdot l_{mix} \left| \frac{dU}{dy} \right|$$

We can say that l_{mix}does not a physical property of fluid and grouping terms, we obtain:

$$\tau_{xy} = \mu_T \frac{dU}{dy}$$

Where $\mu_T$is called eddy viscosity:

$$\mu_T = \rho l_{mix}^2 \left| \frac{dU}{dy} \right|$$

K-Energy Models

Into this point we can include both one equation model and two equation model. The two models keep the Boussinesq approximation to viscosity but these models differ in an important aspect, while the one equations models are incomplete because they link turbulent length scale with a typical dimension of fluid. On the contrary we can observe that the two-equation models have one more equation than one-equation models to establish a turbulent length scale, this is equivalent to present a more complete model.

The equation of turbulent kinetic energy. This equations was developed to incorporate the non local and historical effects of the flow over the eddy viscosity. So Prandtl chose kinetic energy of turbulent fluctuation into different models as basis of velocity of scale.

$$k = \frac{1}{2}\overline{u^2} = \frac{1}{2}(\overline{u_T^2} + \overline{v_T^2} + \overline{w_T^2})$$

So, the eddy viscosity can be established:

$$\mu_T = C \cdot l_{mix} \rho k^{1/2}$$

On the other hand, when we study the equation of Reynold Stresses:

$$\rho\frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \tau_{ij}\frac{\partial U_i}{\partial x_j} - \rho\epsilon + \frac{\partial}{\partial x_j} \cdot \left(\mu\frac{\partial k}{\partial x_j} - \frac{1}{2}\overline{\rho u_{\tau}' u_{\tau}' u_j'} - \overline{p u_j'}\right)$$

We can see several terms which show the processes that occur in a turbulence flow. We may classify into following groups: production terms "k", which show us the percentage released into the turbulence of kinetic energy from main flow. Dissipation terms of "ϵ", which show us the percentage of kinetic energy converted into thermal internal energy.

Two equation models These types of models have been widely studied during 80th and 90th decades, the most of studies made during these years have been made with these models which permit more accuracy into complex flow but they require high computational power. However It isn't any problem because today the computational development is high.

The two equation models provide not only one equation to solve "kinetic energy" but also they give us other equation to solve the mixing length. These models of turbulence permit to predict turbulence flow more accurately. The starting point of these methods we have the Bousinesq approximation together with the equation to kinetic energy. Kolmogorov in 40th decade defined the second variable. This variable was called dissipation "w". This quantity has dimension of t^{-1}, so the values of turbulence length scale, eddy viscosity and turbulence dissipation can be determined by:

$$\mu_T = \tfrac{\rho k}{\omega} \; l \sim \tfrac{k^{1/2}}{\omega} \; \epsilon \sim k\omega$$

We can note that this type of turbulence models is not an universal tool to obtain the turbulence characteristics on the flow, so, depending of their use the results of these will be more or less acceptable.

In the following steps we are going to center into the most utilized two equation model, there is a $k - \varepsilon$ ***model.***

Close models or numeric strategy to solve approximate the Navier - Stokes equations have developed two transport equations more, one to turbulent kinetic energy (k) and the other to turbulent dissipation kinetic energy (ϵ). These variables are defined by following expression:

$$k = \frac{1}{2}\overline{u_t^2} = \frac{1}{2}(\overline{u_t^2} + \overline{v_t^2} + \overline{w_t^2})$$

$$\epsilon = 2v\overline{e_{tj}e_{tj}}$$

where $\overline{e_{tj}}$ is a fluctuating part of strain rate tensor.

The transport equation to "k" and "ϵ" are based in the knowledge of process which produce the changes in these variables and they are:

$$\frac{\partial \rho k}{\partial t} + \nabla(\rho k\overline{\overline{u}}) = 2\mu_T E_{ij}E_{ij} - \rho\epsilon + \nabla\left[\frac{\mu_T}{\sigma_k} grad\, k\right]$$

$$\frac{\partial \rho k}{\partial t} + \nabla(\rho k\overline{\overline{u}}) = C_{1\epsilon}\frac{\epsilon}{k}2\mu_T E_{ij}E_{ij} - C_{2\epsilon}\frac{\epsilon}{k}\rho\epsilon + \nabla\left[\frac{\mu_T}{\sigma_k} grad\, k\right]$$

where E_{ij} is a averaged component of strain rate. The physic meaning of last expression can be summarized into the following balance:

$$[Velocity\, change\, of\, {}^{k}\!/_{\epsilon}] + [{}^{k}\!/_{\epsilon}\, transport\, by\, convection] =$$
$$= [{}^{k}\!/_{\epsilon}\, production] - [destruction\, of\, {}^{k}\!/_{\epsilon}] + [{}^{k}\!/_{\epsilon}\, transport\, by\, diffusión]$$

Here they appeared several kinematic concept connected with scales of typical length, which are associate with flow movement. So, in this turbulence model the turbulence length scale can be shown for the following expression:

$$l = \frac{k^{3/2}}{\epsilon}$$

This method utilizes "ϵ"of little eddies to define length scale "l" of big eddies, because of the extraction energy velocity of big eddies is equal to transfer energy velocity of little eddies into high Reynold numbers. If this weren't so, the energy of certain scales may increase or decrease without limit. This does not ocurr in the reality, so, in this way we can justify to use the dissipation "ϵ" into the length definition "l". Applying the same approximation to mixing length, the eddy viscosity can be obtained like as:

$$\mu_T = C_\mu \rho \frac{k^2}{\epsilon}$$

The two equation models have the next advantages:

- It is only necessary to fix initial and boundary conditions.
- Good results are obtained to great variety of flows.
- Two equation models are the most turbulence methods utilized in the industry applications.
- Development wall laws are disposed to these types of models

On the other hand they also have disadvantages:

- Their implementation is more complex than algebraic models because of introduction of extra differential equations.
- Poor results are obtained in some cases: confined flows, flows with high longitudinal gradients...

A variation of $k - \epsilon$ model, denominated $k - \omega$, was published by Wilkox in 1993. This method removes one variable dividing it all the rest of the problem, simplifying the solving of the problem. This method has the objection, it shows singularities, when the variable which divides the rest of the problem is zero. For this reason $k - \omega$ is more suitable into aeronautic field than into naval field.

Modeling of boundary layer

The boundary layer is part of the fluid field close to solid contour where the viscous phenomenon is shown of specially way. Close to any solid contour a velocity gradient appears in the normal direction of this, due to viscosity and non-sliding condition. The velocity gradient determines the energy exchange between different fluid particles. This phenomenon causes vortex and turbulence.

The basic problem of numerical modeling of the energy exchange in the boundary layer consist into making a correct definition in the velocity of the particles close to boundary. It requires a high density of the mesh. It is necessary to capture the local phenomena which are caused into boundary layer. To make this calculus requires high computational spending. Sometime this isn't possible to make it because of computational limits.

These difficulties have been solved by means of approximations. These approximations will be classified into four groups: models of distributed lost, models of shear layer, models of boundary layer and wall laws.

To establish these approximations, the region close to boundary layer is characterized by dimensionless variables which permit to define the wall conditions. Whereas, these new variables are defined by means of friction velocity terms, which depends on shear stress in the wall.

We call "y" to dimensionless distance to wall and "U" to time averaged velocity, resulting:

$$U^+ = \frac{U}{u_\tau} \qquad y^+ = y \cdot \rho \cdot \frac{u_\tau}{\mu}$$

In this way, we achieve that if the flow depends on the wall condition, we can obtain the universal functions $U^+$and y^+. These functions will be obtained into the limits which permits obtain any characteristics of the boundary layer, in fact it is proved that the relation between these variables exists.

The standard functions of the wall are valid to smooth walls, but they can be modified to include the wall roughness by means of adjusting the constants. If the wall roughness is modeled, the distant to wall is not dimensioned by roughness equivalent height. By this way, some codes include a specific law of the wall to modeled this effect. (*Rough wall field*)

The universal function can be used to link the variables in a first computational mesh. Shear stress can be obtained directly without solving the middle structure.

The standard functions in the *"law of the wall"* are a source of false ideas in the solving of turbulence flows even to expert users in CFD. To obtain a thin layer close to surface is the purpose of these models.

Now, we can note the restrictions when we use the law of the wall:

- The flow solved can agree with the hypothesis made to achieve the law of wall equations.
- It is necessary remark the y^+ into the first points in the mesh to validate the wall functions.
- The functions of the wall do not release of the need to obtain a suitable solving of the turbulence into boundary layer.
- It is difficult to generate a predetermined distribution of y^+, because it depends on the solution of the problem.

Thus, some conditions will be taken into mind in the modeling of turbulent boundary layer:

- Check the lower limit of y^+, which is situated around $y^+ = 20 - 30$
- Check the upper limit of y^+. If the Reynold number is moderate and the boundary layer is situated between $y^+ = 200 - 500$, we can advise that there isn't the opportunity of solving of boundary layer with precision. If the first integration point is placed in value above $y^+ = 100$, this fact occurs.

- Check the solving of boundary layer. If the effects of the boundary layer are important, It is suitable to check the solution of the boundary layer after solving.
- Check the wall functions to verify the roughness in the functions.

9.- Basics techniques of CFD

Boundary Element Methods (BEM)

These techniques are basically employed to solve the potential flows. Thus in several practices hydrodynamics techniques, the BEM methods are called panel methods. In these methods the hull surface is splitted into little panels where each panel satisfies the Laplace equation$\nabla\phi = 0$.

Finite Difference Methods (FEM)

The finite difference methods dominate mainly in the structural calculus. This method occupy a less important role In hydrodynamics calculus. Now we present this method slightly.

The FEM method is a procedure to solve continuous problem by means of dividing the system in a finite number of parts (elements), whose behavior is defined by a finite numbers of parameters. Thus the solving of complete problem is made through assembling the solution of singular elements.

To analyze the behavior of the continuous structure It is necessary to proceed by means of this way:

- The continuous system is divided, by means of imaginary lines and surfaces in a discrete system.
- It is supposed that the elements are connected themselves through a discrete number of points, called nodes which are situated in a boundary. The displacements in nodes will be the unknown factors of the problem, similar to discrete problems.
- It is chosen a bunch of functions which permit to define fluid field in a only way into each element depending of boundary conditions.
- From here the problem is similar to a discrete problem, i.e, the matrix of each element is assembled, the boundary conditions are imposed and the equilibrium equations of fluid field are solved, thus the velocity vectors, stresses,... are determined. However in hydrodynamic several acceptances in elementary calculus and elementary functions can not be used to obtain the error of integrals by the same way of Galerkin methods.

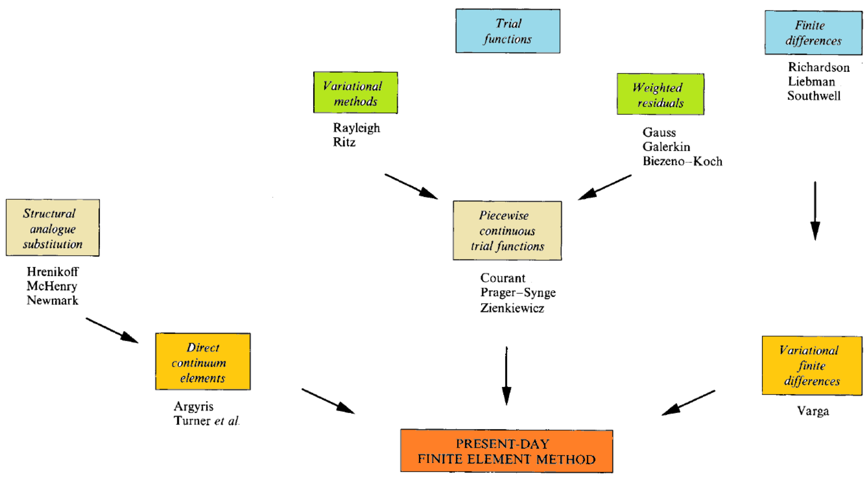
- Then it is determined a system of concentrated forces in a nodes which balances the stresses in the boundary and whatever distributed loads. Thus, it is obtained the relationship between the forces and displacements, which have the following form:

$$q^e = K^e a^e + f^e$$

being $q^e$the vector of forces which act in the nodes, $K^e$the stiffness matrix, $a^e$the vector of nodal displacements, $f^e$the vector of nodal forces to balance the mass forces, the initial stresses, the initial distortions and the stresses applied in a boundaries.

- With the stiffness of each element, the stiffness matrix is assembled. It is applied the boundary conditions and it is established the global equation system. By means of this system they are obtained the global displacements. Finally the stresses are calculated in each element from these displacements.

The method commented is usually called "the displacement method". So, today we can find a variety of approximations to "finite element method" . In the following scheme they are showed different ways and theories to approximate the solution with this method:



Finite Volume Methods (FVM)

This method is similar to the last methods, it is utilized to temporal and spatial discretization of the current problem. However this method integrates the momentum and mass conservation equations over the cell before approximating the value in the central node. Thus the made error in the output face is canceled by the made error in the input face of the cell, this permits to the mass and momentum conservation. Several commercial codes employed this method to solve the equations.

10.- The meshing problems: discretization of the special domain

The codes based in FEM or FVM normally are formed by three parts: pre-processor, calculus module and post-processor. The pre-processor is the responsible of meshing the problem. Thus the meshes of the problem can be classified into several types according to two criteria: the manner of defining the boundary of the scope, and the connectivity between the mesh points.

In terms of how we define the boundaries, they exist the compliant meshes (theses do not interpolate in the boundary definition) and the non compliant meshes (they do not define the real boundary but they define a numerical approximation of the same).

According to the data's structure, they are two methods of mesh generating which create two types of primary meshes: Unstructured and Structured. In the structured meshes, the points are placed according to a family of coordinate lines which permit a direct display the relationship between the points. This fact permits a simplification respect to unstructured mesh.

Whatever the case (may be unstructured or structured), the mesh should achieve of the following features:

- It is necessary to adjust the boundaries of studied regions to achieve the best accuracy in the representation of the boundary conditions.
- The mesh can be distributed locally in a regular way as possible. It has a soft variations of density. The mesh density is defined how the number of points per surface or volume unit. It is basic to select a good mesh density to solve the problem suitably.
- If the mesh is rough, the accuracy in the solving of the problem is not acquired.
- If the mesh is fine, the computational cost probably will be disproportionate.
- The mesh density should be placed in the locations where it will be expected high spatial variations.
- The mesh should adjust dynamically to changes of the flow variables.

The difficulties shown in the mesh generation are usually the following:

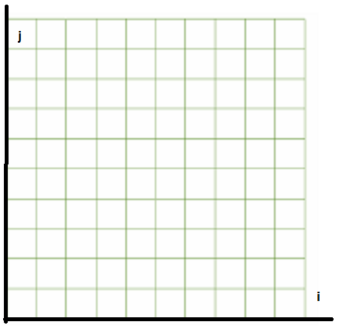
1. *There is a excessive distortion in the elements.* In a Finite Element Method where they are used basic element (uni- bi- and tri- dimensional) with simple forms (lines, triangles, rectangles, cubes...) which are converted in other with arbitrary forms.
2. *There is a incorrect mixed of the element.* Most of programs based in FEM, give to user the facility of choosing the type of element, however all the elements are compatible themselves.
3. *There is a incorrect connection between elements*. There is convergence in FEM whenever C_{m-1} exists continuity, being "m" the functional.

It is basic to choose a suitable mesh density to solve the problem.

The user has to know the response of element and the approximations made in the problem.

The linear elements require finest mesh than quadratic form elements and these require more than the cubic form elements.

STRUCTURED MESH



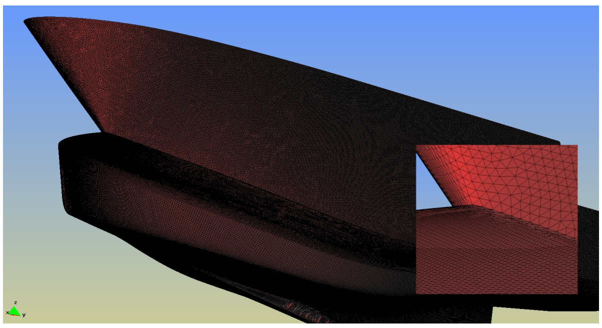
Advantages.

- The structured mesh leads to a simple solving.
- It is required less memory to store the variables.
- The implicit methods utilize structured mesh: ADI, linear relaxation methods, relaxation in the plane...
- The smoothness and orthogonality can be easily controlled.

Disadvantages.

- The use of complex geometries is not easy and flexible.
- The adaptability only is possible through adding or moving lines in a 2D or 3D mesh.
- The boundary's movement is difficult.

UNSTRUCTURED MESH



Advantages

It exists easily to handle complex geometries with adaptability and boundary movement.

The automatic mesh generating is easier, even with 3D complex geometries.

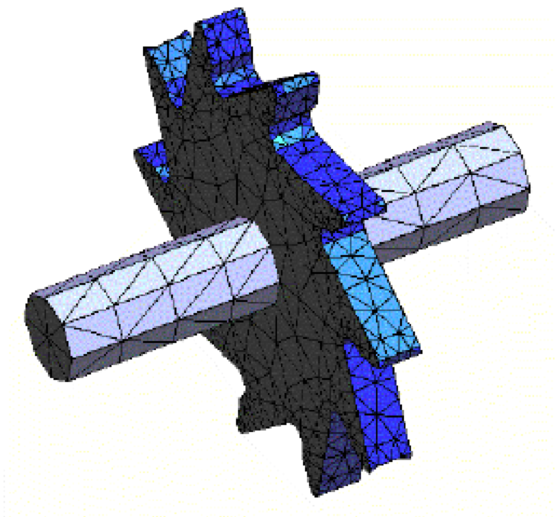
Although we expect a solution with low accuracy, it is not true. We can obtain a high accuracy because the mesh can adapt to solution.

Disadvantages.

It requires a great storage capacity for the variables.

The resolution of the problem is more difficult.

MULTIBODY MESH



We can note that the multibody meshes are generally non compliant, i.e. the nodes do not match in the blocks joints.

Advantages.

- It is possible to handle complex geometries, so, the flexibility increases.
- This permits to block with different sizes.

Disadvantages.

- The mesh generating is usually difficult. The specific way to show boundaries between the blocks in suitable way, requires a high experience in their use.
- This types of mesh are less flexible than unstructured meshes when There is movement in the boundaries.

11.- Basic characteristics of CFD programs.

In this last chapter we are going to deal about what types of interface CFD offers us from a computer user point of view.

Today There is a great variety of CFD codes, from free codes to powerful commercial CFD codes. We can join these codes in some groups:

General Codes: Into these types we can find the codes that have the following tools at the service of the user to: to generate geometries, to apply boundary conditions, to discretise the spatial domain (generally this step is called mesh generating), solve the problem and show the results. Examples of these are: Tdyn, CFX, Fluent, Phoenix, Open FOAM ...

Specific Codes: They are the allocated codes to a concrete part of the CFD application. Into these codes we can find, allocated codes to airfoil design, allocated codes to pneumatic design...

Pre and Post processor: They are allocated codes to mesh generating, to assign boundary conditions, to geometry generating, to take care of specific calculus or allocated codes to show the results. Into the previous examples we should say that the typical CFD codes are those allocate to specific calculus. Similar to general code here there is a great variety of codes:: ParaView, OpenDX , MayaVi Data Visualizer ,GMV (The General Mesh Viewer), FAST, pV3, Visual3 y ChomboVis, VU, CAF2D , FEMFlow and CAF3D, PostFlow, VIGIE, DAVID, Se.La.Vi., HIGHEND (Interactive Graphics using Hierarchical Experimental and Numerical Data), NCSA UNIX Products. . .

Mesh generating codes: They are the allocated codes only to mesh generating. Some examples of them are: ADMesh, ANGENER, AUTOMESH2D, CAF2D GENGRID, CAF2D GENMESH, Cart3D, CGM, Chimera Grid Tools, COG 2. . . .

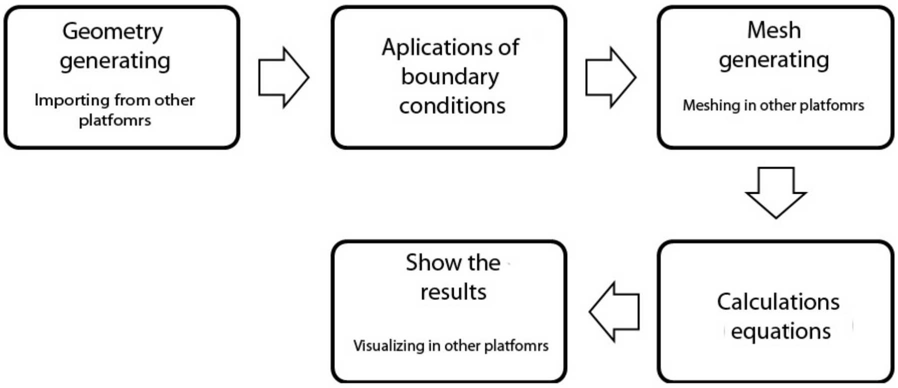
Then, we are going to show the followed process by the user to make an analysis with general CFD codes. Into these types of codes, the experience and knowledge of the user are very important to get a good result in the analysis.

The general codes usually are divided into three parts: preprocessor, a calculus node and postprocessor.

- *The preprocessor* is that part of the CFD codes, allocated to make all previous work necessary to solve the equations of the specific problem.
- *The calculus node* is the part of CFD codes, whose function is to solve numerically the equations placed by preprocessor.

- *The postprocessor* is the other part of the programs, which is allocated to show the results in a graphic form or similar.

Calculus process followed by the user.



The first part of this calculus process consists of geometry generating, in what concerns us, the hull of the ship. It is usually imported from other platforms to CFD codes by means of different exchange files: iges, 3dm, dfx, aciis, parasolid... So, it will only be necessary to create a control volume for solving the studied problem. In our case, in the ship, the control volume should be a parallelogram formed by two parts: one for the water and another for the air.

Then when the user already has a defined geometry with volume control, he passes to assign the boundary conditions necessary to solve the problem. These types of conditions may be Neumann conditions, Dirichlet or mixed. These conditions will be such as: the velocity of the fluid in the boundary, boundary layer, pressure fields or stresses, wave spectrum...

The following step to be made by the user. This step is very important, depending on the accuracy in a spatial discretisation, the results will be more or less accurate. In this step the user creates a mesh over the geometry. If the mesh is very fine in the critical regions of the geometry, the result will be better. However, It is not possible to get the finest mesh, because it depends of the computer capacities. If the mesh is very fine, the calculus will probably last so long.

Then when the problem is meshed, the following step will be the calculus. Finally the user will show the results. In this part the user can obtain a great variety of graphic results from stream lines to interactive videos. Most of codes permit to export the results from these codes to other platforms. By this way the user will can work with the results in other platforms.

Bibliography

[1] R. Ballesteros, J. González, J.M. Fernández, K.M. Argüelles. Técnicas Numéricas en Mecánica de Fluidos. Universidad de Oviedo.

[2] Volker Bertram. "Practical Ship Hydrodynamic." Butterworth Heinemann.

[3] David C. Wilkox. "Turbulence Modeling for CFD". DCW Industries Inc.

[4] Servicio de Apoyo a la Investigación Tecnológica. "El método de los elementos finitos y sus aplicaciones en ingeniería". UPCT

[5] O.C. Zienkiewicz - R.L. Taylor. "El método de los elementos finito. Las bases"- Volumen 1. 5ª Edición. Cimne. Barcelona.

[6] E. Oñate. "Cálculo de Estructuras por el Método de Elementos Finitos. Análisis estático lineal". Cimne. Barcelona.

[7] Presentación ETSIN "Teoría del buque"

[8] Manual Tdyn 6.0. Compass Ingeniería y Sistemas.

[9] Gid, User Manua, Version 8. Compass Ingeniería y Sistemas.

[10] TdynTurbuelenceHB. Compass Ingeniería y sistemas

[11] Departamento de Estructuras y Construcción. "El método de los elementos finitos en ingeniería". 4º Curso de Ingeniería Industrial UPCT.

[12] Luis Pérez Rojas." Los CFD como herramienta en Hidrodinámica". AULAS DEL MAR. UPCT

[13] L. Pérez Rojas. A. Souto Iglesias. P. Roca Fernández. "Los CFD en la formación hidrodinámica del ingeniero naval". IV SIMPOSIUM PANAMERICANO DE EDUCACIÓN EN INGENIERÍA NAVAL.

[14] J. Valle Cabezas. "Ayer, hoy y mañana de la Hidrodinámica numérica". conferencia pronunciada con motivo de la celebración del 75 aniversario del CEHIPAR.

[15] http://www.cfd-online.com