

Saw-tooth Approximation and Numerical Solution for Flows in Inclined Channels

CHRISTINA G. GEORGANTOPOULOU

School of Engineering
Bahrain Polytechnic
PO Box 33349, Isa Town
Kingdom of Bahrain
christina@polytechnic.bh

GEORGE A. GEORGANTOPOULOS

Hellenic Air Force Academy
Dekeleia Attikis
GREECE
gageorgant@yahoo.gr

NIKOLAOS S. VASILIKOS

School of Pedagogical and Technological Education
Iraklion Attikis
GREECE
vasnikoler@yahoo.gr

Abstract: - Incompressible flows inside short or extended pipes present multiple industrial applications and their numerical study is essential for the related companies in order to control the flow variables. The present work is a flexible numerical approach and solution for incompressible flows inside inclined channels. Despite of the non-Cartesian channel bound, a saw-tooth Cartesian grid generation methodology is applied in combination with a finite volume scheme in order to predict the flow field as well as the recirculation zones of the flow. The new Cartesian approximated bound is consisted of only grid lines while solutions are provided for laminar viscous flows for various Reynolds numbers. The solution of Navier-Stokes equations is based on an artificial compressibility technique in combination with the flux-vector splitting methodology. We prove that the methodology is independent of the grid size, step angle or expansion ratio of the channel and the final results present satisfied accuracy despite of the geometry approximation which is taking place during the mesh generation. The utility of the algorithms is tested using related results using body fitted grid approach as well as the flow simulation software FLUENT. We conclude that the below method is appropriate for industrial flows study, manly in power industries, reducing the computational time and providing a simple and flexible scheme which easily can be used and modified by engineers without advanced knowledge in CFD.

Keywords: - Cartesian grids, inclined pipes, saw-tooth method, incompressible flows, sub – grids, viscous flows.

1. Introduction

The various industrial flows inside pipes present significant interest due to control variable issues, deduction of friction, reducing turbulence or specify the recirculation zones. Unlimited number of applications is developed at Middle East Countries as at the Kingdom of Bahrain where the power industries presence is quite extended. Although all the related companies use computational codes in order to control the pipeline networks, the need of a

simple and flexible code for the numerical simulation and study of the pipes' flows is highly demanded, providing the ability to industrial engineers appropriate modify the geometries or optimize the flows.

Various CFD approaches have been presented concerning channels' flows with many numerical approaches and applications. Torres [1] is presenting an accurate model for recalculating flows inside pipes, while many methodologies have been

developed for flows over backward facing steps [2, 3, 4, and 5]. For all the researchers and engineers is important to investigate specific regions in curved pipes or in particular non-straight pipes' parts as at Louda's [6] work for branched channels, Liu's one [7] for grooved channels, or Marn's paper [8] for the laminar flow in a 90° pipe bend. Nobari has presented an interesting numerical study for stationary and rotating pipes [9], straight or curved. The sudden expansion which is developed to backward facing steps, not only in perpendicular cases but in inclined also present major interest in industrial applications [2, 10].

Most of the above approaches use body fitted curvilinear or hybrid grids except of the cases of straight or backward facing steps (angle=90°) pipes where Cartesian grids are developed. This type of grids presents better flexibility and provides easy specification of the geometry description, which is very important for industrial flow applications. Interesting approaches are Coirier's [11] who is used a Cartesian methodology for steady transonic solutions Euler's equations and in [12] performed accuracy and efficiency assessments of the method. It's a cell-centred method with an interesting treatment of boundary conditions. Additionally, Smith [13], develops a grid generation procedure that uses Cartesian embedded unstructured approach for complex geometries and prove that the accuracy. Especially at the case of numerical simulation for pipelines, regardless the curvilinear bounds which exist through any network, the Cartesian grid option seems to be the most appropriate.

As we meet at the literature, numerous computational approaches have been developed for pipes including inclined ones, which are common used in industries, like power plants, shipping repairing and maintenance, aluminium production etc. Louda [14], has done a very interesting work concerning inclined step short pipes using body fitted un-uniform grids, presenting very satisfied results regarding the flow as the analysis of the recirculation zones. Kozel [15,16], presents finite volume and finite elements methodologies applied in back facing and incline step flows laminar and turbulent respectively. Many other researchers have studied the above cases giving emphasis in recirculating zones according to the step's angle, working fluid as well as the aspect ratio of the channel.

The present work is a part of a research effort in order to mathematical simulate and estimate industrial flows inside various pipes for industrial applications in power industries. The major target is to prove that this Cartesian grid methodology is

appropriate and accurate enough in order to predict any industrial flows, regardless the viscosity, type of flow or flow rate, as well as regardless any geometrical characteristic of the pipe. The grid generation methodology is based on a previous work in Cartesian grid generation for curvilinear geometries which has successfully predicted various incompressible flows despite the approximated bound which has to be produced in internal as well as in external flows [17,18]. In order to overcome the huge computational memory which in some flow cases is developed, we apply a block nested refinement technique based on the sequence of sub-grids [18]. A first approach for flows inside channels with only Cartesian bounds has already been presented providing a simple, flexible and accurate numerical solution for pipe flows capable to be used for industrial applications in power industries [19,20]. In this work we extend the methodology in pipes where the physical outer bound will be replaced by an approximated Cartesian one. The N-S equations are solved using finite volume analysis and the artificial compressibility technique in combination with flux-vector splitting methodology [21]. The flow which are studied are laminar and incompressible for various Re numbers giving emphasis in flow variables and the prediction of recirculating flows. Particular investigation is taking place concerning the appropriate angle of the step in order to produce the desired expansion with the minimum recirculation simultaneously.

2. Numerical Model Analysis

The numerical approach of flow fields consists of several steps, where plenty of methods can be applied. In order to simulate and estimate the aforementioned incompressible flows we follow the below major steps:

- Physical domain discretization
- Cartesian grid generation
- Governing equation discretization
- Boundary Conditions application.
- Numerical solution and results processing.

According to the case more steps may be developed in order to solve the flow problem with the desired accuracy [22,23]. Especially at the cases of inclined steps a precise way of geometry description is needed in order to ensure the flow rate conservation and predict the recirculation lengths as well as the important points with accuracy. The method which

we propose is independent of the physical geometry description and it can be applied even if a complex curvilinear bound has to be approached. At the following sections each step of our numerical methodology is presented although further analytical details can be found in previous published works [18, 19, 20].

2.1 Cartesian bound creation

In our numerical approach, if the physical domain bounds are aligned with Cartesian grid lines, we have no need to produce any approximated bound. However, if our domain consists of complex or curvilinear bounds, we have to create a new approximated one in order to proceed to the grid generation.

At the case of incline steps most of the outer physical domain bound are aligned with Cartesian grid lines except one (2D - incline step line, fig. 1) which has to be approached due to the nature of our methodology. The new approximate bound is parted only by the use of grid lines, on x or z-axis either and this is our benefit. The method is used, called saw-tooth and is has been chosen as the most appropriate for the finite volume cell centered numerical simulation of flow fields. This method provides independence and automation of grid generation for problems with complex boundaries, with or without existence of an analytical function. The new geometrical approach is based on sets of data points, the original points as well as the approximated points of the body contour as we describe below.

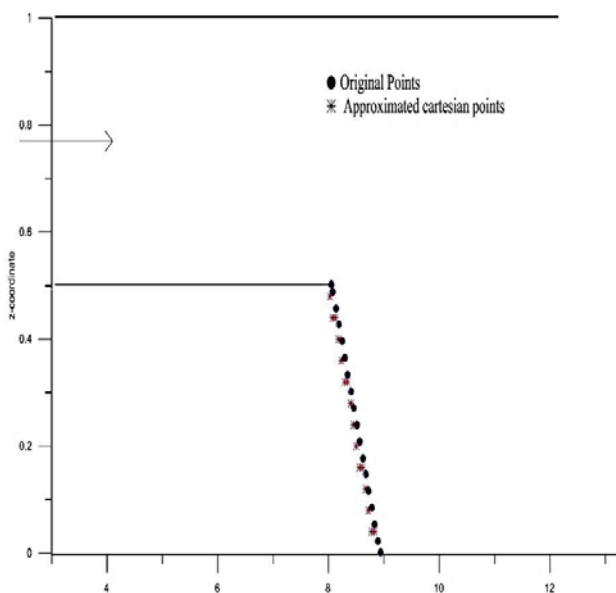


Fig. 1: Part of the inclined channel; original and approximated points of the outer bound.

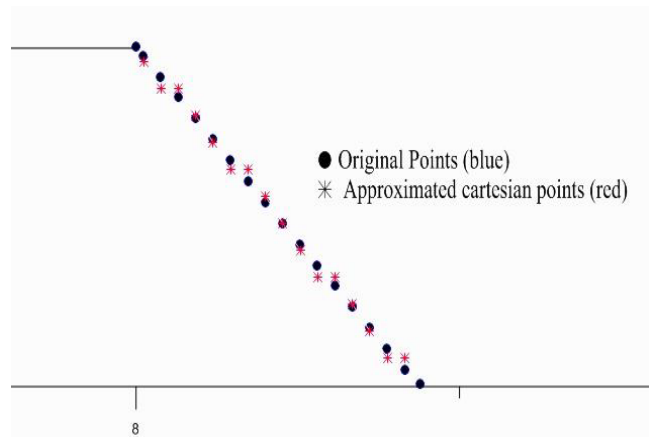


Fig.2: Original and corresponding approximated points at the inclined step

Consequently, the first step is the creation of a new approximated Cartesian bound of our physical domain. We project the original contour of the curvilinear geometry onto a Cartesian grid. This complex contour is described by a set of data points on x or z-axis either (blue points, figure 2). We have to control if the contour segment between two neighbour data points varies monotonically with respect to both x or z directions [20]. In order to define the new approximated points we follow the below rule: if an original data point is on x-axis, we calculate the distance between this and its neighbouring grid nodes in the same direction (x). According the smallest distance we choose the corresponding grid node as the Cartesian approximated point, (fig. 2).

Let's assume that $(Ox(i), Oz(j))$ is a point of the original bound lying on a grid line with x constant. In order to define which one is the corresponded grid point and the minimum distance we calculate the followings:

$$d_1 = |Oz(j) - z(k)| \tag{1}$$

And
$$d_2 = |Oz(j) - z(k + 1)| \tag{2}$$

where $z(k)$ and $z(k+1)$ are neighbouring grid nodes lying on a x-constant grid line. We have controlled also that:

$$z(k) \leq Oz(j) \leq z(k + 1) \tag{3}$$

By this way we define the new points, applying the rule of minimum distance for each set of original data points. We finally connect the new points applying saw-tooth method, using only grid lines (figure 4).

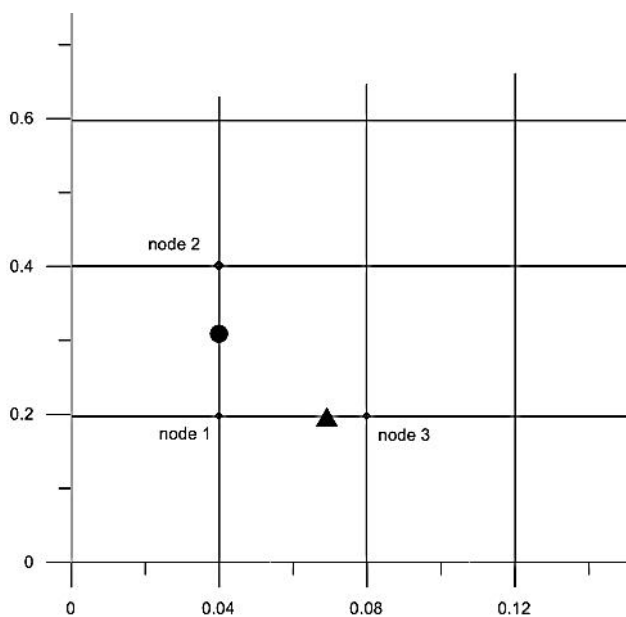


Fig. 3: Definition of the approximated Cartesian points. According to the rule of minimum distance the original data will be replaced of Cartesian grid nodes.

2.2 Mesh generation and refinement technique

After the above procedure we have defined points on the Cartesian grid nodes which are described the outer bound of the physical domain. At this time the Cartesian grid generation is simple, excluding the cells that there are no more included to the flow field due to the above approximation. Therefore a new Cartesian grid is generated where all the “new physical” bounds lie on grid lines.

Although for the numerical solution of an inclined step pipe, a uniform Cartesian grid can produce accurate results, a refinement technique can be also applied in order to reduce the computational memory as well as the CPU time. We apply a block nested methodology by the use of a hierarchical structured grid approach. The method is based on using a sequence of nested rectangular meshes in which numerical simulation is taking place (fig. 5). The whole domain is a rectangle whose sides lie in the coordinate directions except the case of including the approximated Cartesian outer bound. Then only three edges are straight lines while the fourth one is a crooked line, like in the case of the inclined pipe. We simulate the domain based in as many refine grids as we need [23].

There is no specific rule concerning the location of the nested sub-grids. We choose the number and the

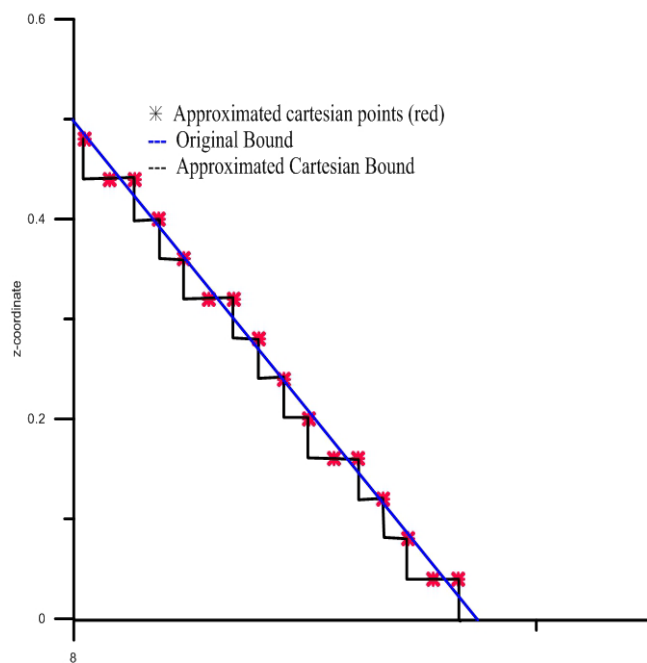


Fig. 4: Method for the connection of the Cartesian approximated points (saw-tooth)

exact position according to the flow application and if it is possible, after we apply a numerical simulation and solution using uniform Cartesian grids. By this first solution we can conclude concerning the critical parts of the domain where refinement needed to be applied.

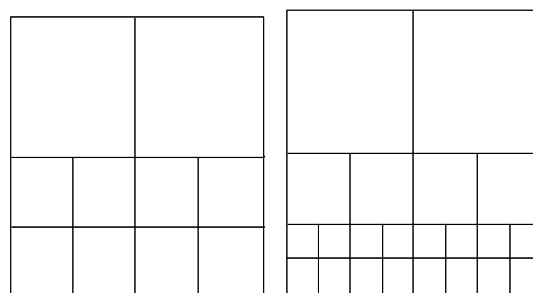


Fig. 5: Block nested refinement, grid levels=2 and refinement factor = 2

At the case of the inclined pipes, two are the most important reasons which guide us to the final position of the sub-grids as well as the level of the refinement:

- The approximated outer bound; we need a refinement technique in order to reduce the relative error [18].

- The possibility of a recirculation zone development, mainly after or on the step, according to the Reynolds number [14].

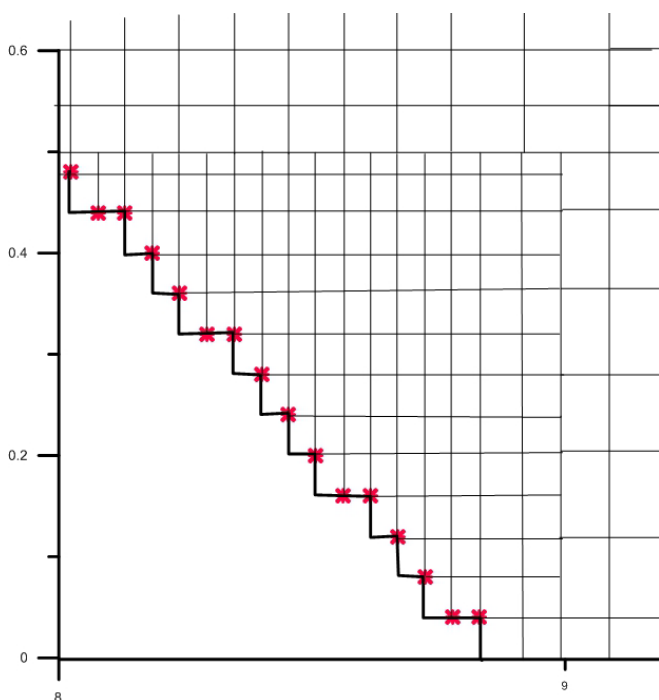


Fig. 6: Block nested refinement technique for the inclined short pipe, one level of refinement.

Especially in extended pipeline networks, low interest is presented at the straight parts of these, where the domain discretization can be occurred using uniform Cartesian grids developing the minimum number of cells. At the regions with curvilinear bounds, as at the case of the inclined channels, the block nested refinement technique can be easily applied providing better accuracy and reducing the computational time simultaneously. In this method the refinement is convenient to be a power of 2.

The proposed nested algorithm contains several levels of grids. We name the coarsest level $m=0$ and each next refine sub – grid is named $m+1$. [24]. We define an integer refinement factor:

$$I = dx_m / dx_{m+1} = dz_m / dz_{m+1}. \quad (4)$$

As we have created the coarse grid we simulate the flow field and calculate the variables. We have already defined the limits of the refinement levels as well as the value of the refinement factor and we proceed the calculation to the next refinement level. The sub-grids bounds must lie on a grid line of the previous level grid. As we use staggered grids and

the variable values are expressed on the cell's center, we consider artificial cells, all around the physical domain and the sub – grids too, in order to apply the boundary conditions. By this way we estimate the variables using interpolation between pseudo – cells and their neighbor cells for the velocity value. As we have fulfilled the simulation in all sub-grids and we have the flow field results at m_{\max} level, we resolve the problem in the coarser levels again to ensure conservation. We find a new solution, this time by the influence of the fine levels. In addition we must satisfy both Dirichlet and Neumann matching conditions along coarse-fine and fine- coarse interfaces. That's why we give the velocity values, but we solve for pressure. With nested grids, each grid is separately defined and has its own solution vector, so that a grid can be advanced independently of other grids, except for the determination of its boundary values. The information exchange between two successive levels is described in the next section.

This refinement methodology is not necessary for the inclined step pipe numerical estimation, but it will provide accurate results in a reduced computational time. It worth to be mentioned that the above refinement technique is automatic and it is developed giving only the refinement factor as data.

2.3 Boundary Conditions

There are two different cases for the boundary conditions application according to the aforementioned methodology, at the inclined pipes. The first one is regarding the boundary conditions at the physical bound and the second between coarse – fine interfaces.

Concerning the boundary conditions at the outer geometry bound, we apply Dirichlet and Neumann matching conditions, according to the bound following the direction that can be seen at figure 7. The boundary conditions' application at the interfaces follows either the linear interpolation rule for the velocity values, either the definition of the pressure vertical derivative. We represent $u^{m+1}(i,k)$ and $w^{m+1}(i,k)$, the values of the velocity components on the sub-grid pseudo-cells. The $u^m(l,n)$ and $w^m(l,n)$ are the corresponding coarse grid values into the physical domain. At the time that we solve at the m grid level first and we proceed to the $m+1$ sub-grid the boundary conditions for the axial velocity component values, if we consider that we apply these along x -axis, are given as below:

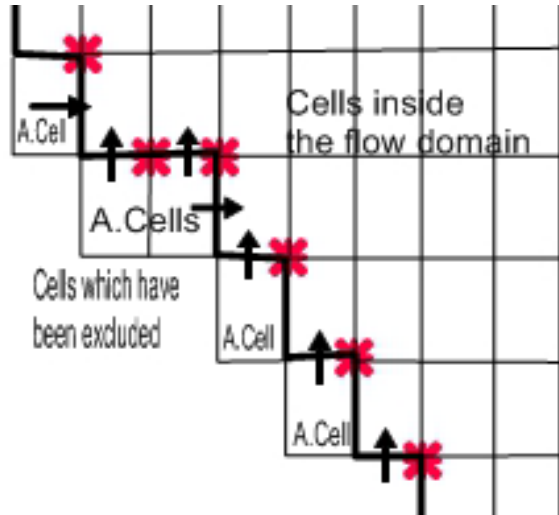


Fig 7: Various ways of boundary conditions application according to the outer Cartesian bound shape. A.cells=artificial cells.

$$u^{m+1}(i, k) = u^{m+1}(i, k + 1) = u^m(l, n) \quad (5)$$

And if we apply along z-axis:

$$u^{m+1}(i, k) = u^{m+1}(i + 1, k) = u^m(l, n) \quad (6)$$

Regarding the case that we have to transfer the values between refine – coarse interfaces along x-axis again, we apply:

$$u^m(l, n) = \frac{u^{m+1}(i, k) + u^{m+1}(i, k + 1)}{2} \quad (7)$$

And

$$w^m(l, n) = \frac{w^{m+1}(i, k) + w^{m+1}(i, k + 1)}{2} \quad (8)$$

In all the above formulas the refinement factor has been set to be equal to two (2).

Concerning the pressure boundary condition, we prefer not to apply interpolation and develop a different approach for this variable. Assuming that we simulate for an axisymmetric flow, the pressure vertical derivative at the interface is estimated as follows at equation 9.

In fact, we can apply liner interpolation for both of the above flow variables with satisfied results. In our case we prefer to create an non - depended technique and a robust solution which will be freely created through our flow field.

$$\frac{\partial p}{\partial n} = n_x \left[\frac{1}{\text{Re}} \left(\frac{\partial^2 u}{\partial y^2} + \frac{1}{y} \cdot \frac{\partial u}{\partial y} + \frac{\partial^2 u}{\partial x^2} \right) - u \cdot \frac{\partial u}{\partial x} - v \cdot \frac{\partial u}{\partial y} \right] + n_y \left[\frac{1}{\text{Re}} \left(\frac{\partial^2 v}{\partial y^2} + \frac{1}{y} \cdot \frac{\partial v}{\partial y} + \frac{\partial^2 v}{\partial x^2} - \frac{v}{y^2} \right) - u \cdot \frac{\partial v}{\partial x} - v \cdot \frac{\partial v}{\partial y} \right] \quad (9)$$

That's why we believe that we have to solve for the pressure as many researchers do [24]. This approach has been successfully applied to a wide range of various fluid or heat transfer applications [25,26,27,28].

2.4 Numerical model of the governing equations

Our mathematical modeling numerically solves the incompressible governing flow equations which are the Navier-Stokes equations, and are expressed in terms of the Cartesian system of coordinates (x, z) , as below:

$$[\Gamma] \frac{\partial q}{\partial t} + \frac{\partial e}{\partial x} + \frac{\partial g}{\partial z} + \alpha \cdot \frac{g_1}{z} = \frac{1}{\text{Re}} \left(\frac{\partial r}{\partial x} + \frac{\partial s}{\partial z} + \alpha \cdot \frac{s_1}{z} \right) \quad (10)$$

Where,

$$[\Gamma] = \text{diag} \left(\frac{1}{\beta}, 1, 1 \right)^T \quad (11)$$

after the addition of the artificial compressibility term β , α is a switch for the activation of the axisymmetric terms ($\alpha=0$ is non axisymmetric, $\alpha=1$ axisymmetric flow field), Re the Reynolds number and Q the unknown solution vector, $Q = (p \ u \ w)^T$, with p being the pressure, and (u, w) the velocity components in physical space. E, G, G_1 and R, S, S_1 are respectively the convective and diffusive flux vectors at the plane (x, z) .

The above N-S equations are the governing equations for unsteady flow. These are also used for the solution of steady flow fields as in our case. In these steady cases the time derivative is used for the construction of an iterative technique using the artificial time step in order to define our final steady state. Here, we extend the FVS method for solving incompressible flow fields implicitly. In such flow fields the splitting of the convective flux vectors has to change sense because of their non-homogeneous property. The values of the flux vectors at the cell

faces are approached by upwind schemes up to third order of accuracy. The unfactored discretized Navier-Stokes equations are solved by an implicit second order accurate in time scheme, using Gauss-Seidel relaxation technique [27].

3. Results

We present the numerical simulation and calculation of the incompressible, laminar and steady flow inside an inclined step short pipe using the above mathematical scheme. We explore our Cartesian grid generation efficiency in the above flow problem and we try to define the velocity and pressure distribution along the channel. The calculation have been repeated for various grid sizes, uniform or refine nested types of grids, various refinement factors as well as for many different Reynolds values. In order to validate our results, some cases have been numerical simulated and estimated through the flow solver FLUENT. The comparison of the results were satisfied enough proving the independence of our methodology.

3.1 Flow inside an inclined step pipe

The numerical estimation of an inclined step pipe provides valuable information concerning the flow variables distribution as well as the recirculation zone data along the pipe. The methodology which has been presented is independent of the grid size and refinement factor as well as the angle of the step. At the specific test case the angle of the step is equal to 30° , while 8 lengths have been set before the step and 20 lengths after that in order to fully develop the flow. (Figure 8). The main reason that this test case has been chosen is not only the appearance of an interesting industrial flow (power plants, heat exchangers applications) but also the close relationship of the specific case with the numerical modelling of the air in an urban environment which we intent to develop in our future research, and by this way we can validate our methodology. However this flow appears detachments and reattachment points as well as recirculation zones and boundary layers which vary according to the aspect ratio, the expansion ratio or the Reynolds number of the flow.

The expansion ratio is equal to 0.5 while all the variables are estimated with the reference length to be equal to the diameter of the cylinder. ($L_{ref}=H$)

The dimensionless pressure \tilde{p} is defined as below:

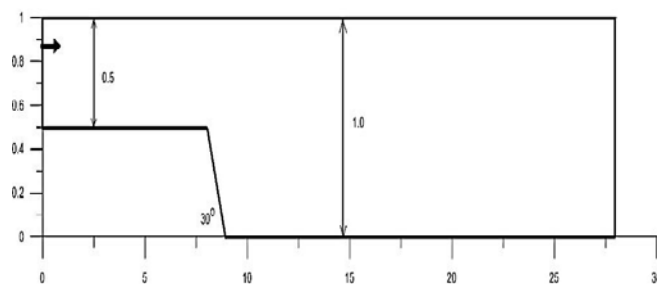


Fig. 8: Physical domain of the inclined step channel.

$$\tilde{p} = \frac{P}{\rho U_{ref}^2} \quad (12)$$

where P the pressure, ρ the density of the fluid and U_{ref} , the reference velocity which is equal to the average velocity to the inlet of the channel.

The grid generation and the numerical method that was described above were used for the calculation of the flow inside the short pipe. Several numerical approaches have been developed using uniform grids and block nested ones. The following results are extracted using base grid size 521×26 , uniform and refined with level=1 and refinement factor $I=2$. (figures 9,10). The artificial compressibility factor is set equal to 1 as optimum choice for the reduction of the computational time [18], while the axisymmetric term equal to 0 as the flow problem has been addressed as 2-dimensional. The Re number, that was based on the maximum inlet velocity and the diameter of the inlet, was set equal to 100, 400 as well as to 800.

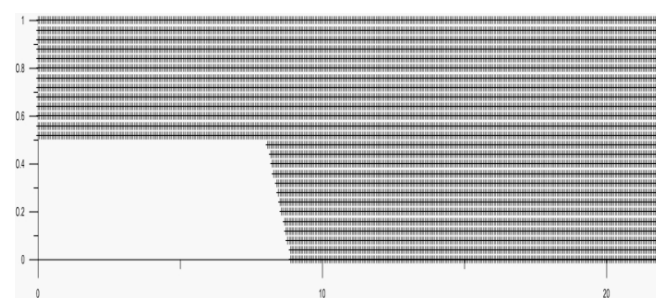


Fig. 9: Parts of the uniform Cartesian grid 521×26 for the numerical estimation of the inclined step channel.

The boundary conditions are summarized as below, at table I. It is worth to be mentioned that due to the below outlet boundary conditions, we need to choose the appropriate length of the cylinder; 15

dimensionless lengths for $Re=100$ and 30 ones for $Re=400, 800$. In order to control the accuracy of the proposed method, we simulated the current flow field by the use of un-uniform body fitted grid, sized using FLUENT software applying the same conditions for the flow fields.

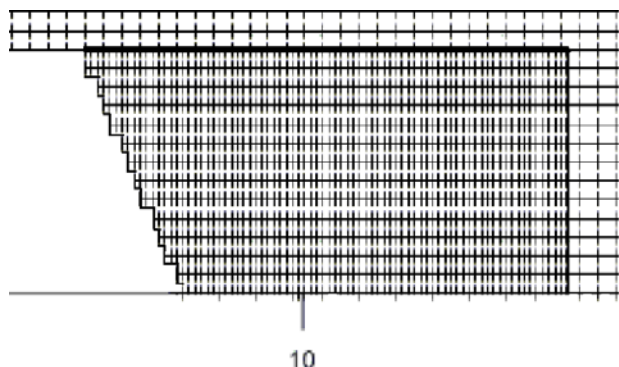


Fig. 10: Part of the used block nested grid, $I=1$ and $I=2$ around the step. Base grid size 211×11 .

Table I: Boundary Conditions for the channel's simulation

<u>Upper bound.:</u>	Wall conditions: $u = w = 0, \frac{\partial p}{\partial z} = 0$
<u>Lower bound.:</u>	Wall conditions: $u = w = 0, \frac{\partial p}{\partial z} = 0$
<u>Inlet:</u>	Inlet conditions giving the value of the velocity: $u = 1, w = 0, \frac{\partial p}{\partial x} = 0$
<u>Outlet:</u>	Outlet conditions where the pressure has a value: $\frac{\partial u}{\partial x} = 0, w = 0, p = 0$

At the above table u and w are the axial and vertical components of the velocity of the fluid respectively, p is the pressure and x, z are the Cartesian coordinates. As we can see at the table, at the outlet boundary we give the pressure while at the inlet the axial velocity is given.

In order to validate our results, we develop the aforementioned flow field using FLUENT software, 521×26 grid size, applying the same conditions as these have been described above.

We present the axial velocity profiles for various positions along the channel at the figures 11, 12 below with very satisfied convergence. Although the recirculation is low for $Re=100$, it can be detected as presented from both of the numerical methods (u profile, $x=9$).

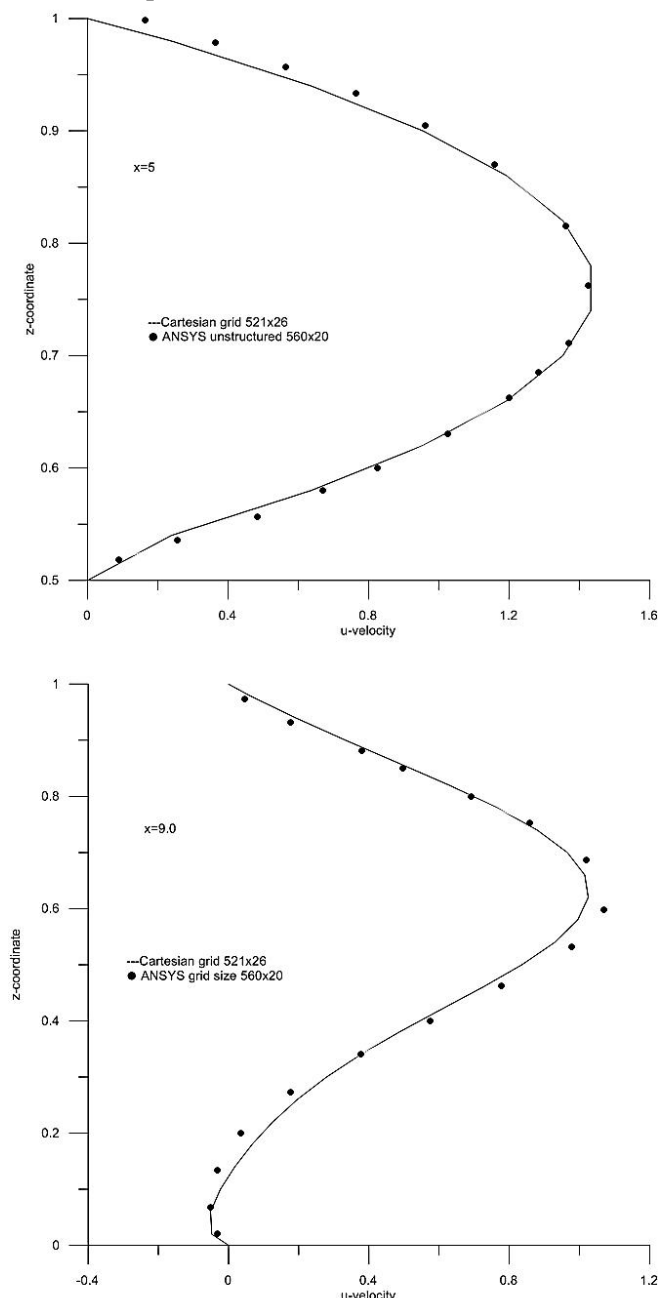


Fig. 11: Velocity profile along channel with inclined step angle $=\pi/6$, $Re=100$ on $x=5$ (before the step) and $x=9$ (after the step). For the Cartesian refined methodology based grid $211 \times 11, I=2$.

It's worth to be mentioned that despite of the slight differences which are depicted to the above figures, the mass flow rate is concerned in both of the numerical estimations.

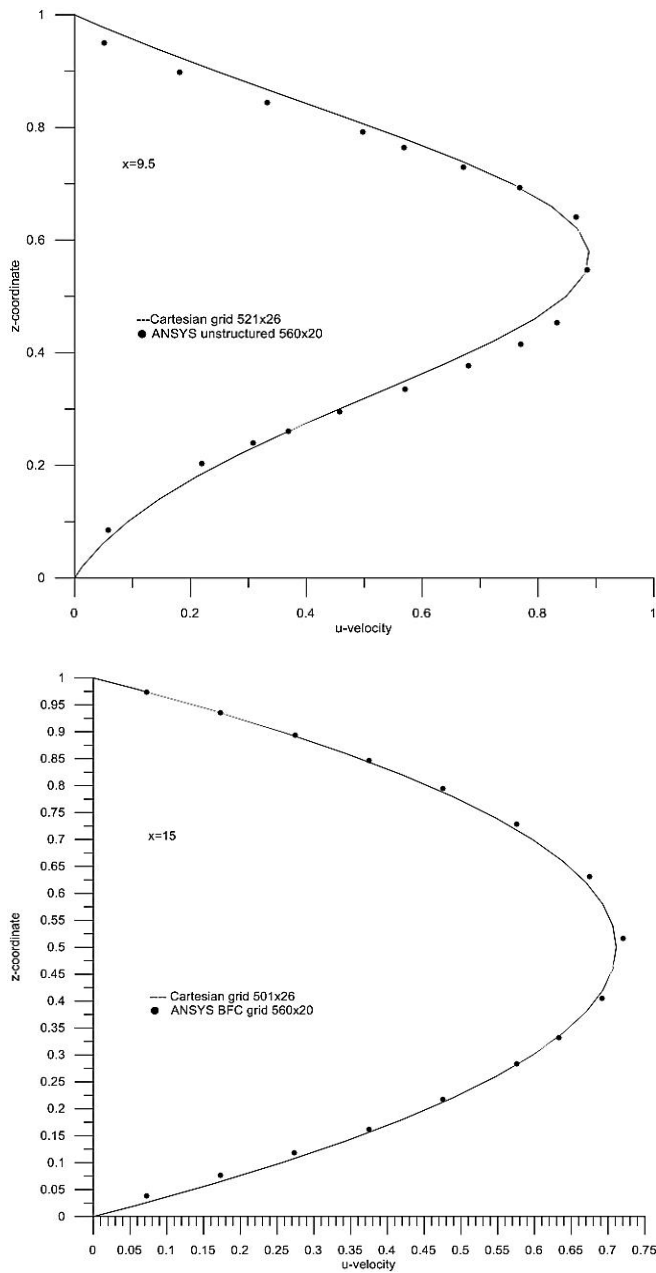


Fig. 12: Velocity profile along channel with inclined step angle= $\pi/6$, $Re=100$ on $x=9.5$ and $x=15$ (after the step). For the Cartesian refined methodology based grid 211×11 , $I=2$.

We also present the pressure distribution produced by the Cartesian – artificial compressibility method at figure 13 as well as by FLUENT software at figure 14. The convergence of the pressure results seems to be very satisfied, following the previous boundary conditions which have been set. No

problems have been occurred either due to the approximated Cartesian bound, either to the nested block sub grids.

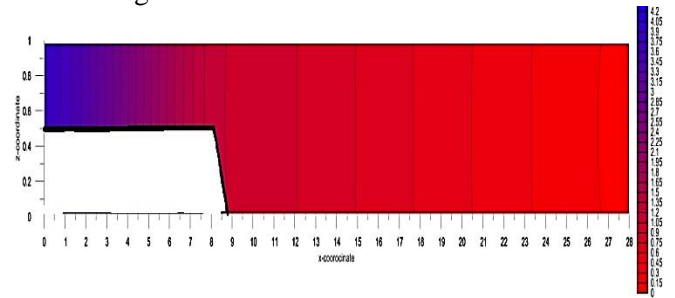


Fig. 13: Pressure distribution along the channel. Cartesian refined methodology with based grid size 211×11 , $I=2$.

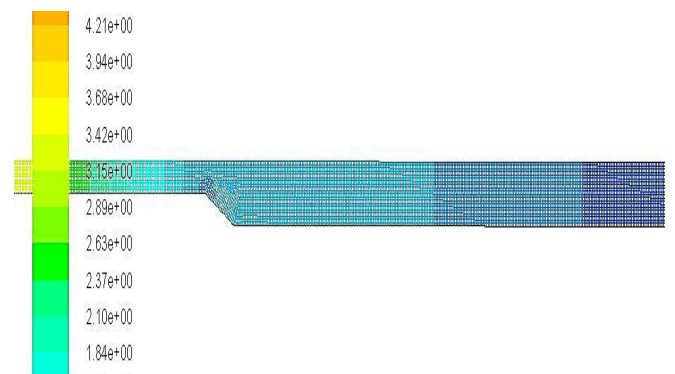


Fig. 14: Pressure distribution along the channel using FLUENT. Body fitted grid, size 521×26 .

Due to the fact that we have managed to estimate the recirculation area (low intense) in both of the cases with high accuracy, it seems that the application of the boundary conditions is appropriate, especially on the approximated Cartesian bound. Good behavior is also provided through the neighboring nested block grids. The velocity vectors are presented at figure 15, close to the incline step, providing a clear picture of the recirculation along the flow field.

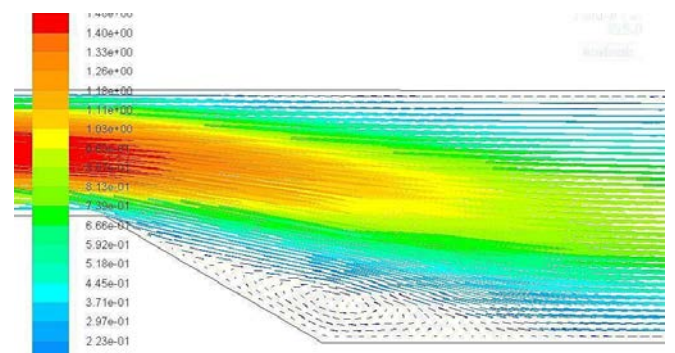


Fig. 15: Velocity vectors along the channel. FLUENT, grid size 521×26 , $Re=100$.

4. Conclusions

A numerical simulation and estimation of the flow inside incline step channel for incompressible flows is presented. Concerning the discretization of the physical domain we apply a saw-tooth method while the final approximation of the geometrical bound is taking place by the use of only Cartesian grid lines. We generate uniform as well as refined Cartesian grids using block nested structured sub-grids, where the numerical approach demands. We use a cell center discretization and the boundary transfer is demonstrated in the interfaces by the use of interpolation for the velocity and pressure values at the coarse – fine interfaces of the refined sub – grids. The method is applied for steady, laminar, viscous and incompressible flows. We pay attention at the approximated bound of the physical domain, in order to check the accurate prediction of the recirculation

We present the numerical solution using a refinement block nested technique and we validate our methodology with the corresponded results of FLUENT software with very satisfied convergence. The velocity as well as the pressure values seems to be appropriate, while a low intense recirculation is detected after the step. By these results it seems that using the inclined channel with angle up to 30o the flow presents appropriate and quite good behavior without major recirculation zones or friction problems as these have been presented to the corresponded channel with angle equal to 90o [20]. By the other hand the Cartesian refined algorithm produces accurate results, despite the approximated bound, providing a flexible numerical approach for a variety of industrial applications. By the use of the refinement technique the accuracy of the numerical results is very satisfied, reducing the computational time and memory simultaneously. With appropriate choice of local block refinement multilevel solutions computed with this algorithm can attain the accuracy of the equivalent uniform fine grid at less computational cost (figures 11, 12). The Cartesian block nested methodology is simple, grid independent and it can be applied in any channel flow regarding the possible non-Cartesian bound. It seems that it can be a useful tool for the prediction of industrial flows as these are developed in power industries.

Acknowledgments:

The authors appreciate the contribution of Mr. Ehab Ali, and Mr. Feras Ali, Bahrain Polytechnic students, for an initial numerical approach of the aforementioned flow using FLUENT and for the treatment of some final FLUENT plots respectively.

References:

- [1] Torres Maj., Garcia J., Numerical characterization of particle dispersion in the turbulent recirculation zones of sudden expansion pipe flows, Proceedings of the 6th European conference on computational Fluid dynamics, Barcelona, 2014
- [2] Santos R., Oliveira K., Figueiredo J., Influence study of the entrance channel in a two-dimensional backward-facing step flow, *Mecanica Computacional*, vol. XXIX, pp. 3347-3358, 2010
- [3] Lee T. and Mateescu D. Experimental and numerical investigation of 2-D backward-facing step flow. *Journal of fluids and structures*, vol. 12, pp. 703–716, 1998
- [4] Armaly B., Durst F., Pereira J., and Schounung B. Experimental and theoretical investigation of backward-facing step flow. *Journal of fluid mechanics*, vol. 127, pp 473–496, 1983.
- [5] Manzan W., Vilela C., Mariano F., “Experimental and computational simulations of the flows over backward facing step”, Proceedings of 22nd Int. COBEM2013, SP, Brazil
- [6] Louda P., Kozel K., Prijoda J., Benes L., Kopacek T., Numerical solution for incompressible flow through branched channel”, *Comput. Fluids*, vol. 43, pp. 268-276, 2013
- [7] Wallin S., Johansson A.V., “ A complete explicit algebraic Reynolds stress model for incompressible and compressible turbulent flows”, *Journal Fluid Mechanics*, vol. 403, pp.89-132
- [8] Liu C., Liu Z., McCormic S., “Multilevel adaptive methods for incompressible flow in grooved channels”, *J. of Computational and Applied Mathematics*, vol. 38, pp. 283-295, 1991
- [9] Marn J., Ternik P., “Laminar flow of a shear thickening fluid in a 90o pipe bend”, *Fluid Dynamics Research*, vol. 38, pp. 295-312, , 2006
- [10] Nassab S., Moosavi R., Sarvani S., “ Turbulent forced convection flow adjacent to inclined forward step in a duct”, *Int. J. of Thermal Sciences*, vol. 48, pp. 1319-1326, 2008
- [11] Coirier, W.J. and Powell, K.G., “Solution-Adaptive Cartesian cell approach for viscous and inviscid flows”, *AIAA J.*, Vol. 34, pp. 938-945, 1996

- [12] Coirier, W.J. and Powell, K.G., "An accuracy assessment of Cartesian-mesh approaches for the Euler equations", *J. of Computational Physics*, Vol. 117, pp. 121-131, 1995
- [13] Smith, R.J. and Johnston, L.J., "A novel approach to engineering computations for complex aerodynamic flows", *Proceedings of the 4th International Conference on Numerical grid Generation in Computational Fluid Dynamics and related Fields*, pp. 271-285, 1994
- [14] Louda P., Prihoda J., Kozel K., Svacek P., "Numerical simulations of flows over 2D and 3D backward facing inclined steps", *Int. Journal of heat and Fluid flow*.
- [15] Kozel K., Louda P., Ptihoda J., *Numerical Solution of 2D and 3D backward facing steps*, *PAMM 5*, vol 1, pp. 467-468
- [16] Kozel K., Louda P., Svacek P., Prihoda J., *Finite volume and finite element methods applied to backward faciong steps*, *Int. Conference from Scientific Computing to Computational Engineering*, Patras.
- [17] Georgantopoulou Chr.G., Pappou Th.J., Tsaggaris S.G., "Cartesian grid generator for N-S numerical simulation of flow fields in curvilinear geometries", *Proceedings of the 4th GRACM congress on comput. Mechanics*, pp. 526-534, 2002
- [18] Chr.G. Georgantopoulou and S. Tsangaris, "Block mesh refinement for incompressible flows in curvilinear domains", *Applied Mathematical Modeling*, Vol.31, pp2136-2148
- [19] Chr.G.Georgantopoulou, G.A.Georgantopoulos and S.Tsangaris, "Incompressible navier stokes equations solution using block nested Cartesian grid", *Proceedings of 25th ICAS2006*, Humburg.
- [20] Georgantopoulou C., Vasilikos N., Georgantopoulos G., "Mathematical modelling for the solution of incompressible flow through channels using block structured grids", *WSEAS MAMECTIS14*, Lisbon, Portugal
- [21] Pappou, Th. and Tsangaris, S., "Development of an artificial compressibility methodology using Flux Vector Splitting", *International J. for Numerical Methods in Fluid*, Vol. 25, pp.523-545, 1997
- [22] Yu K., Lau K., Chan C., Zhang H., "2D and 3D computation for two phase flow by large eddy simulation and a lagrangian model", *Proc. Of the 3rd IASME/WSEAS*, Corfu, Greece
- [23] Shamloo H., Pirzadeh B., "Investigation of characteristics of separation zones in t- junctions", *Proc. Of Int. conference WSEAS on Water resources, Hydarulics and Hydrology*, pp. 157-161, UK
- [24] Martin D. and Collela P., "A cell-centered adaptive projection method for the incompressible Euler equations.", *J. of Computational Physics*, Vol. 163, pp. 271-312, 2000
- [25] Mavromatidis L., Michel P., Mankibi M. and Santamouris M., "Investigation of the sensibility of Multi-Foil insulations using the guarded hot plate and the guarded hot box test methods", *Palenc 2010, 5th Eur. Conf. on energy Performance (EPIC2010)*, 29 Sep.-1Oct, Rhodes island, Greece, 2010
- [26] Mavromatidis L., Michel P., Mankibi M. and Santamouris M., "Investigation of the contribution of multifoil insulation in the veduction of energy consumption of buildings", *9th Geographical Conf.*, 4-6Nov, Athens, Greece, 2010
- [27] Georgantopoulou Chr., Georgantopoulos G., Vasilikos N. and Tsangaris S., "Cartesian refinement grid generation and numerical calculation of flows around Naca0012 airfoil" *Proceedings of Int. Conference on Applied mathematics, simulation, modeling*, pp.256-263
- [28] Berger M.J. and Collela P., "Local adaptive mesh refinement for shock hydrodynamics", *J. of Comput. Physics*, Vol. 83, pp.64-84, 1989.
- [29] Pappou, Th. and Tsangaris, S., "Development of an artificial compressibility methodology using Flux Vector Splitting", *International J. for Numerical Methods in Fluid*, Vol. 25, pp.523-545, 1997