CODEN STJSAO ISSN 0562-1887
ZX470/1400 UDK 532.51:536.247:519.61/.64

Applications of Adaptive-mesh Refinement in Computational Fluid Dynamics to Problems of Turbulent Flow and Heat Exchange

Franjo JURETIĆ[1)] **and** ***David GOSMAN***[2)]

1) AVL-AST d.o.o., Avenija Dubrovnik 10/3, HR - 10020 Zagreb, **Republic of Croatia**

2) CD-Adapco Ltd., 200Shepherds Bush Road, London W6 7NL, **United Kingdom**

franjo.juretic@avl.com

Original scientific paper

Most processes occurring in devices in air-conditioning systems require fluid flow and heat exchange, and the functioning of these devices is crucially dependent on their understanding. Computational Fluid Dynamics is a modern tool which enables development engineers to simulate the physics of processes on a computer. Such simulations may contain errors because the original problem has been replaced by a simplified discrete problem, solved on a mesh of non-overlapping control volumes and assuming certain solution behaviour in every control volume of the modelled domain. This paper presents the advantages of adaptive-mesh refinement which helps engineers get accurate solutions without their intervention by modifying the mesh where higher accuracy is needed. The potential of the method is shown on some examples often found in engineering practice.

Primjena automatskog rafiniranja računalne mreže u Numeričkoj mehanici fluida na problemima turbulentnog toka i izmjene topline

Izvornoznanstveni članak

Strujanje fluida i izmjena topline su važni procesi u klimatizacijskim uređajima, stoga je njihovo funkcioniranje vrlo vezano uz razumijevanje procesa strujanja i izmjene topline. Numerička mehanika fluida (CFD) je moderni alat koji inženjerima omogućuje simuliranje fizike procesa na računalu. No, takve simulacije mogu sadržavati grješke zato jer se sustav parcijalnih diferencijalnih jednadžbi zamjenjuje s pojednostavljenim diskretnim problemom, koji se rješava na mreži ne-preklapajućih kontrolnih volumena, uz pretpostavku ponašanja rješenja u kontrolnom volumenu domene. U ovom članku iznesene su prednosti lokalnog rafiniranja računalne mreže koji omogućava inženjerima dobivanje točnih rješenja matematičkog modela automatski i bez njihove intervencije kroz automatsku modifikaciju računalne mreže na mjestima gdje je potrebna veća točnost. Mogućnosti metode su prikazane na primjerima prisutnima u inženjerskoj praksi.

Keywords
Adaptive-mesh refinement
Computational Fluid Dynamics (CFD)
Discretisation Error Estimation

Ključne riječi
Procjena grješke diskretizacije
Računalna mehanika fluida (CFD)
Rafiniranje računalne mreže

Received (primljeno): 2009-04-20
Accepted (prihvaćeno): 2009-08-31

1. Introduction

The modern era of strict ecological regulations imposes many requirements on the quality of the processes occurring in almost all products aimed to the mass market. These requirements together with the pressure of short time to market have forced scientists and engineers to introduce computer-aided simulation tools into the development process, which help them gain insight into the physics of the process occurring in the device under consideration.

Computational Fluid Dynamics (CFD) simulates fluid flow and heat exchange on a computer. The simulation process consists of the following steps:

1. Selection of mathematical model relevant for the physics under consideration. The mathematical model is a system of partial differential equations describing the underlying physics. However, most mathematical models are merely a simplified description of physics thus causing the so-called modelling error defined as the difference between

Symbols/Oznake

$\vec{u}$	- velocity vector, m/s - vektor brzine
p	- pressure, Pa - tlak
$\vec{g}$	- acceleration of the gravity force, m/s^2 - ubrzanje sile teže
ρ	- density, kg/m^3 - gustoća
c_p	- specific heat capacity at constant pressure, J/(kg·K) - specifični toplinski kapacitet pri konstantnom tlaku
λ	- heat conductivity, W/(m·K) - toplinska provodnost
R	- specific gas constant, J/(kg·K) - plinska konstanta
r	- radius, m - radius
d	- diameter, m - promjer
T	- temperature, K - temperatura
μ	- dynamic viscosity, Pas - dinamička viskoznost
μ_t	- turbulent viscosity, Pas - turbulentna viskoznost
G_r	- Grashof number - Grashof značajka
N_u	- Nusselt number - Nusseltova značajka
Re	- Reynolds number - Reynoldsov broj

Indices/Indeksi

avg	- averaged value - srednja vrijednost
1	- inner sphere - unutarnja kugla
2	- outer sphere - vanjska kugla

the solution of the model and the exact behaviour of the system [1].

2. Discretisation of model equations. In this study, the Finite Volume Method (FVM) is used, see [1, 2]. The method requires a mesh of non-overlapping control volumes, which determine the points where the solution is sought. The result of this process is a system of algebraic equations, one per control volume, which can be solved on a computer. Naturally, the number of the control volumes in the mesh has an impact on the accuracy of the approximation. The difference between the exact solution of the mathematical model and the approximated solution is called discretisation error [1]. The focus of this study is the reduction of this type of errors, which can be achieved by refining the mesh in the regions of high errors in the approximated solution. This process is called adaptive-mesh refinement, and it consists of error estimation in the approximated solution, serving as an indicator where the mesh shall be changed in order to achieve required accuracy.

Adaptive-refinement strategies can be grouped into four main groups, depending on the type of changes imposed on the mesh and/or the discretisation practice, thus [3]:

- ***h*-refinement** is a strategy where additional computational cells are inserted in regions of high error. Cells can also be removed from regions of small error using a coarsening procedure. The number of cells may increase or decrease during the calculation process as a consequence of additions and removals. The topology of the mesh changes during the adaptation process to achieve a mesh with optimum cell distribution. Examples can be found in Chen *et.al.* [4], Chang *et.al.* [5], Mavriplis [6], Almeida *et.al.* [7], Apel *et.al.* [8] and many more.
- ***r*-refinement** keeps the number of cells and their topology constant, and changes their distribution in order to minimise the error. However, with this approach there is no guarantee that the required accuracy can be achieved with the specified number of cells. The approach may also cause severe distortion of the mesh and convergence problems. Examples of this approach can be found in Ait-Ali-Yahia *et.al.* [9], Gnoffo [10] and others.
- ***p*-refinement** refers to the group of methods which change the order of the discretisation practice according to the error distribution. This approach is particularly suitable for Finite Element calculations as the nature of the discretisation practice allows easy implementation of higher-order methods. The *p*-refinement method is not suitable for non-smooth solutions as higher-order methods are more prone to non-physical oscillations in regions of

steep gradients than their lower order counterparts. Calculations using p-refinement have been presented by Zienkiewicz *et.al.* [11], Demkowicz *et.al.* [12], *etc.*

- **Composite methods.** These methods combine previously mentioned methodologies, see Tam *et.al.* [13] and many more.

Although adaptive methods have evolved during the past few decades, none can generate high-quality anisotropic meshes, required to achieve high accuracy with the smallest possible number of cells, and achieve it in a small number of adaptation cycles. This paper presents an h-refinement technique, because the authors believe that h-refinement provides the best control of the mesh resolution and quality. It is also considered to be the most suitable for problems of engineering interest where the geometries are complex.

2. Governing equations

This study focused on steady-state problems of Newtonian fluids. For this class of problems the mathematical model consists of the following equations:

$$\nabla \cdot (\rho \vec{u}) = 0, \tag{1}$$

$$\begin{aligned} &\nabla \cdot (\rho \vec{u}\vec{u}) - \nabla \cdot (\mu (\nabla \vec{u} + \nabla \vec{u}^{T})) = \\ &= -\nabla p - \nabla (\frac{2}{3}\mu \nabla \cdot \vec{u}) + \rho\, \vec{g}, \end{aligned} \tag{2}$$

$$\nabla \cdot (\rho c_{\mathrm{p}} T \vec{u}) - \nabla \cdot (\lambda \nabla T) = \nabla \cdot (p \vec{u}) - p \nabla \cdot \vec{u}. \tag{3}$$

The system is closed with the constitutive relation for the ideal gas:

$$p = \rho R T. \qquad (4).$$

The system is solved with OpenFOAM, an open-source toolbox for Continuum Mechanics. The code uses a second-order accurate Finite Volume Method (FVM) [2] in a segregated manner, equation-by-equation. The analysis of the discretisation errors for all terms in the above equations is given in [3].

For turbulent incompressible flows without the heat exchange, the system can be further simplified to:

$$\nabla \cdot \vec{u} = 0 \qquad (5),$$

$$\nabla \cdot (\rho \vec{u}\vec{u}) - \nabla \cdot ((\mu + \mu_{\mathrm{t}}) \nabla \vec{u}) = \frac{1}{\rho} \nabla p, \qquad (6),$$

where μ_{t} is a turbulent viscosity calculated from the turbulence model selected for the calculation. The solution procedure modifies the continuity equation into the equation for pressure in order to find the pressure for which the equations are satisfied [1-2].

3. Adaptive-mesh refinement methodology

The mesh-refinement procedure consists of the following steps [3]:

1. Solve the problem on the available mesh.
2. Estimate the distribution of the discretisation error. The error is estimated on every face of the mesh by using the Face Residual Error Estimator [3]. The estimator provides the level of discretisation error for every face, which is used to drive the mesh refinement. In case the error is below the required level on every face, the calculation is terminated.

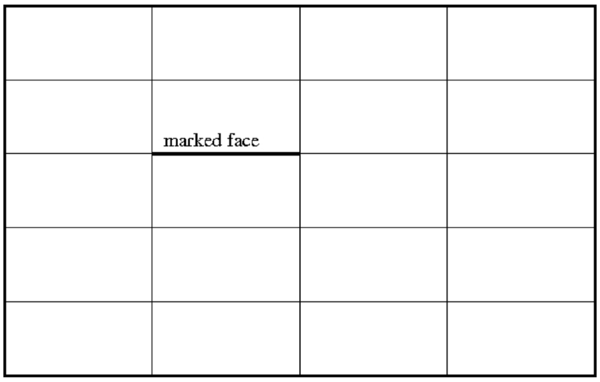

Figure 1. Mesh with marked face

Slika 1. Mreža s označenom stranicom

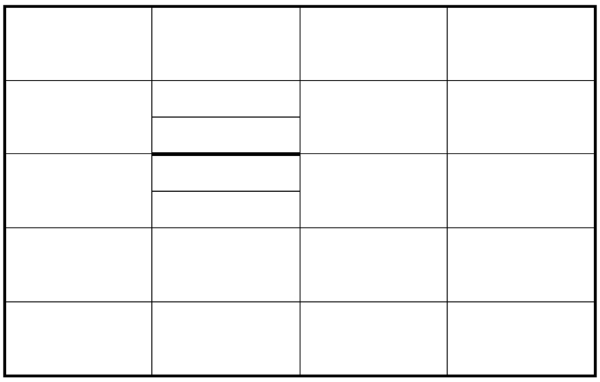

Figure 2. Refined mesh

Slika 2. Rafinirana mreža

3. Determine faces for which the estimated error is above the required upper limit and mark them. Cells sharing marked faces are marked for refinement parallel to those faces, see Figure 1, to reduce distances between the nodes which mainly contribute to the error on that face, see Figure 2.
4. In order to ensure the correct boundary description, newly created boundary vertices may have to be moved from the position where they were located by the refinement procedure to the closest point at the boundary surface.

5. Transfer the solution from the initial mesh onto the new one. This step is implemented in order to reduce the overall calculation time.
6. Repeat from step 1 until the error is reduced below the required upper limit.

4. Results

In order to demonstrate the potential of the method, two cases have been selected. The first problem is an example of turbulent flow across a bank of staggered tubes and the second problem is a natural convection between the two concentric spheres at different temperature.

4.1. Tube bundle

The flow across a bank of staggered tubes is an example of incompressible turbulent flow at constant temperature, governed by equations 5 and 6. The turbulence is modelled by using a q-ζ model by Gibson et.al. [14]. The geometry of the computational domain and the boundary conditions employed here are shown in Figure 3. Even though the flow is periodic, see Figure 5, periodic boundary conditions were not employed for the inlet and the outlet boundary, because the refinement procedure cannot guarantee that the adapted mesh will have the same distribution of faces at the left and the right boundary, as required by the flow solver.

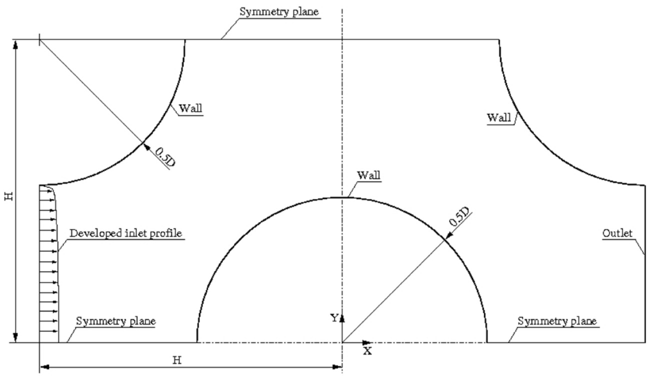

Figure 3. Geometry and boundary conditions for the tube bundle case

Slika 3. Geometrija i rubni uvjeti za primjer snopa cijevi

The Reynolds number of the flow is $Re = 18000$ based on the tube diameter D and the average inlet velocity U_{avg}. A developed inlet profile is obtained by fitting a curve through the data obtained by a separate calculation using periodic boundary conditions.

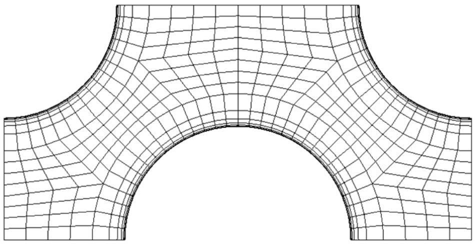

Figure 4. Starting mesh for the tube bundle case (640 cells)

Slika 4. Početna mreža za primjer snopa cijevi (640 ćelija)

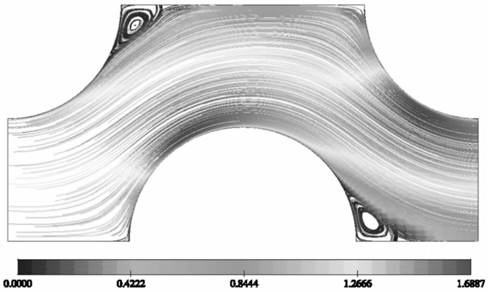

Figure 5. Streamlines of the velocity field for the tube bundle case

Slika 5. Strujnice polja brzine za primjer snopa cijevi

The calculation was started on the coarse mesh with 640 cells, shown in Figure 4. Figure 5 shows the streamlines of the flow which is periodic with a separation on the leeward side of each tube.

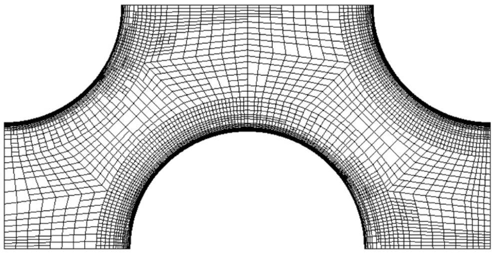

Figure 6. Refined mesh for the tube bundle case (17505 cells)

Slika 6. Rafinirana mreža za primjer snopa cijevi (17505 ćelija)

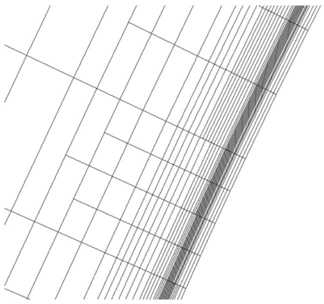

Figure 7. Detail of the refined mesh for the tube bundle case (17505 cells)

Slika 7. Detalj rafinirane mreže za primjer snopa cijevi (17505 ćelija)

The required accuracy of the adapted solution was set to 1 % of the maximum value found in velocity, pressure and turbulence fields. The quality of the solution on the adapted mesh will be compared to the benchmark solution on the mesh consisting of 486000 cells considered discretisation error free. Since the variation of velocity is sharp in the boundary layer region, the mesh is refined primarily in the boundary layer region in the direction parallel to the wall, as shown in Figure 6 and Figure 7.

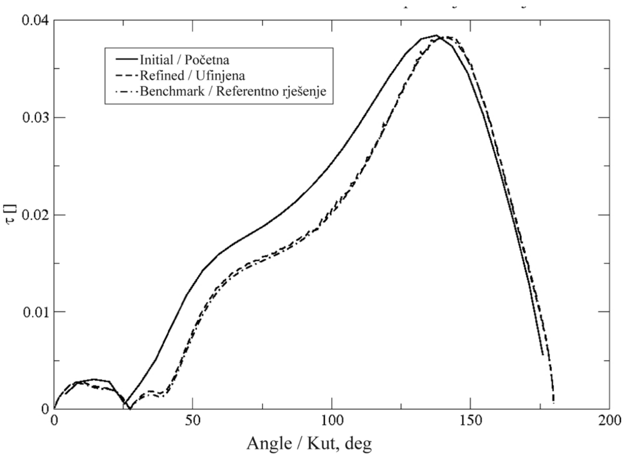

Figure 8. Shear stress at the bottom tube wall

Slika 8. Smično naprezanje na stijenci donje cijevi

The accuracy was also monitored in terms of shear stress at the wall boundaries because it is critically dependent on the mesh resolution in the near-wall region. Figure 8 shows the shear stress at the bottom tube wall, given in a non-dimensional form:

$$\tau_{\mathrm{w}} = \frac{\int \mu \nabla \vec{u} \cdot d\vec{n}}{\frac{1}{2}\rho u_{\mathrm{avg}}^{2}}. \tag{7}$$

It compares favourably with the benchmark solution even though the adapted mesh has only 3.6 % of the number of cells of the benchmark mesh. Calculation time and required computational resources are also low compared to the calculation time and memory resources needed for the benchmark solution.)

4.2. Concentric spheres

The problem of heat transfer and fluid flow in two concentric spheres is an example of flow driven by the natural convection. The geometry of the domain and the boundary conditions are given in Figure 9.

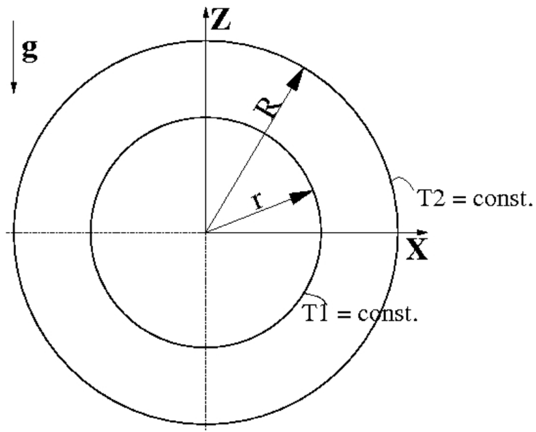

Figure 9. Geometry and boundary conditions for concentric spheres

Slika 9. Geometrija i rubni uvjeti za primjer koncentričnih kugli

The radius of the inner sphere is r_1 = 5 cm, and the radius of the outer sphere is r_2 = 15 cm. The temperature of the inner sphere is T_1 = 373 K and the temperature of the outer sphere is set to T_2 = 293 K. The working fluid is air at atmospheric pressure 101325 Pa. This results in the Grashof number $Gr = 2.44 \cdot 10^7$.

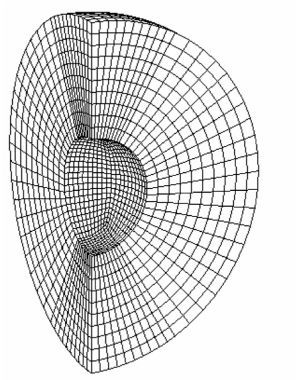

Figure 10. Starting mesh for the concentric spheres (6000 cells)

Slika 10. Početna mreža za primjer koncentričnih kugli (6000 ćelija)

The starting mesh for the calculation is given in Figure 10. It is coarse and consists of 6000 cells; a small number for a 3D problem.

Figures 11 and 12 show the temperature and the velocity field, respectively. The temperature is high near the inner sphere and in the plume rising towards the colder outer sphere. The velocity is also highest in the plume. This indicates that natural convection is the main transport mechanism.

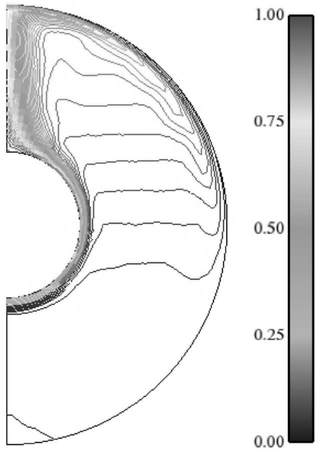

Figure 11. Non-dimensional temperature field for the concentric spheres

Slika 11. Bezdimenzionalno polje temperature za primjer koncentričnih kugli

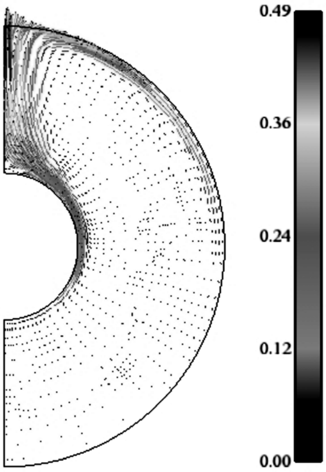

Figure 12. Velocity field for the concentric spheres, m/s

Slika 12. Polje brzina za primjer koncentričnih kugli, m/s

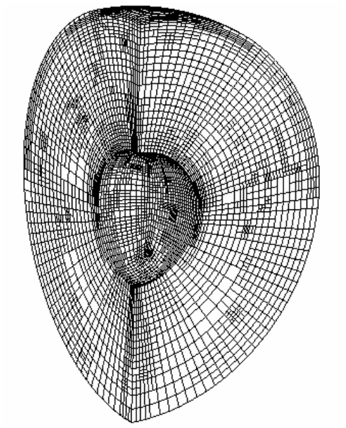

Figure 13. Refine mesh for the concentric spheres (70192 cells)

Slika 13. Rafinirana mreža za primjer koncentričnih kugli (70192 ćelija)

The required accuracy was set to 3 % of maximum values found in the velocity, temperature and pressure fields. Figure 13 shows that the majority of refinement occurred in boundary layer regions on the upper half of the inner and outer sphere. These are the regions of sharp variations of temperature and velocity in the direction normal to the wall, which cannot be resolved accurately on a coarse mesh. The mesh is also randomly refined in some regions, and it is believed that this was caused by the fluctuations of the solution, because the problem is not steady. Since the error estimation and mesh refinement are performed only at certain times, these random refinements exhibit the features of the solution at those time steps.

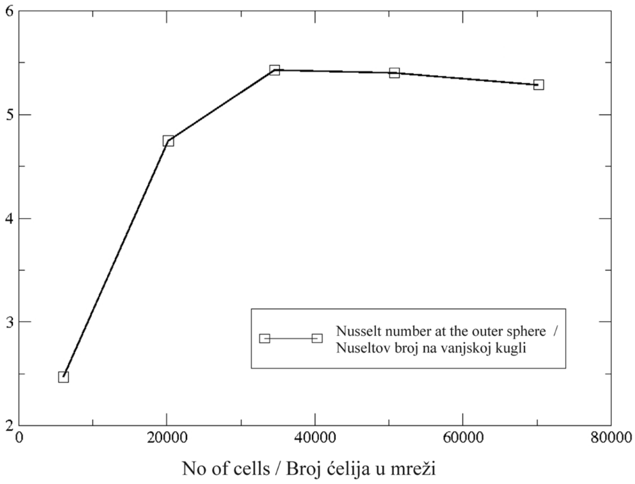

Figure 14. Variation of the Nusselt number with mesh refinement for the concentric spheres

Slika 14. Promjena Nusseltovog broja s rafiniranjem mreže za primjer koncentričnih kugli

The Nusselt number was selected as the main criterion for monitoring the quality of the solution with mesh refinement. Figure 14 shows the values of Nusselt number at the outer sphere on 5 consecutive meshes during the calculation. Please note the dramatic change of the predicted Nusselt number between the first mesh with 6000 cells and third mesh with 34525 cells. The calculation on the initial mesh under-predicted the value by more than 50 % which is not acceptable from an engineering point of view. The variation over the last three meshes is quite small even though the resolution of the mesh changed quite significantly. This indicates that the solution is close to mesh-independence and that the remaining discretisation error is within the 3 % threshold.

5. Summary and conclusions

The paper has presented an anisotropic method for adaptive-mesh refinement which was applied to two cases of interest to the engineering community. The method is found capable of improving accuracy of solutions with reasonable computational resources due to anisotropic refinement of cells. Additionally, the method can be further improved by performing error estimation and refinement based on averaged fields in case of transient cases.

REFERENCES

[1] FERZIGER, J.H.; PERIĆ, M.: *Computational methods for fluid dynamics*, Springer Verlag, Berlin-New York, 1995.

[2] JASAK, H.: *Error analysis and estimation in the Finite Volume method with applications to fluid flows*, PhD Thesis, Imperial College, University of London, 1996.

[3] JURETIĆ, F.: *Error Analysis in Finite Volume CFD*, PhD Thesis, Imperial College, University of London, 2005.

[4] CHEN, W.L.; LIEN, F.S.; LESCHZINER, M. A.: *Local mesh refinement within a multi-block structured-grid scheme for general flows*, Comput. Methods Appl. Mech. Engrg, Vol. 144, pages 327-369, 1997.

[5] CHANG, S.; HAWORTH, D. C.: *Adaptive grid refinement using cell-level and global imbalances*, Int. J. Numer. Meth. Fluids, Vol. 24, pages 375-392, 1997.

[6] MAVRIPLIS, D. J.: *Adaptive Mesh Generation for Viscous Flows Using Delaunay triangulation*, J. Comput. Phys., Vol. 90, pages 271-291, 1990.

[7] ALMEIDA, R. C.; FEIJOO, R. A.; GALEAO, A. C.; PADRA, C.; SILVA, R. S.: *Adaptive finite element computational fluid dynamics using and anisotropic error estimator,* Comput. Methods Appl. Mech. Engrg., Vol. 182, pages 379-400, 2000.

[8] APEL, T.; GROSMAN, S.; JIMACK, P. K.; MEYER, A.: *A new methodology for anisotropic mesh refinement based upon error gradients,* Appl. Numer. Math., Vol. 50, pages 329-341, 2004.

[9] AIT-ALI-YAHIA, D.; HABASHI, W. G.; TAM, A.; VALLET, M. G.; FORTIN, M.: *A directionally adaptive methodology using and Edge-Based Error Estimate on quadrilateral grids,* Int. J. Numer. Meth. Fluids, Vol. 23, pages 673-690, 1996.

[10] GNOFFO, P. A.: *A Finite-Volume, Adaptive Grid Algorithm Applied to Planetary Entry Flowfields*, AIAA J., Vol. 21, pages 1249-1254, 1983.

[11] ZIENKIEWICZ, O. C. and TAYLOR, R. L., *The Finite Element method,* vol 1: Basic formulation and linear problems, McGraw-Hill, 4th edition, 1989.

[12] DEMKOWICZ, L.; ODEN, J.T.; RACHOWICZ, W. and HARDY, O.: *Toward a universal h-p adaptive Finite Element strategy: Part 1: Constrained approximation and data structure,* Comput. Methods Appl. Mech. Engrg., Vol. 77, pages 79-112, 1989.

[13] TAM, A.; AIT-ALI-YAHIA, D.; ROBICHAUD, M. P.; MOORE, M.; KOZEL, V. and HABASHI, W. G.: *Anisotropic mesh adaptation for 3D flows on structured and unstructured grids*, Comput. Methods Appl. Mech. Engrg., Vol. 189, pages 1205-1230, 2000.

[14] GIBSON, M. M.; DAFA'ALLA, A. A.: *Two-Equation Model for Turbulent Wall Flow,* AIAA J., 33(1995), 1514-1518.