A numerical investigation of testing the performance of three different types of meshes and selecting the best type for staggered tube banks.

Abstract:

Three different types of meshes were carried out to test the accuracy and the whole performance for a complicated curved geometry; i.e. staggered configuration in a bank of tubes in laminar flow using STAR CCM+ to execute the flow simulation. The three meshes are named; block structured quadrilateral cells, unstructured grid quadrilateral cells (PAVE) and TETRA mesh made of triangular cells. However, their results have been compared with a result of a very fine grid. Some flow characteristics were selected in a form of contours to show the performance of different meshes includes; velocity and pressure distributions through the flow domain. In addition to that, the velocity profiles at the symmetry line (y=0) and at the periodic boundaries (x=0) were also tested in order to show the closest mesh to the very fine mesh. The best mesh was the block mesh which is very close to the very fine grid and presented better results of flow characteristics such as velocity profile, pressure drop coefficient (Kp) and recirculation length. Moreover; for turbulent flow using a standard k- ε model, the investigation of how the pressure drop coefficient changes with Reynolds number and the relation between recirculation length versus Reynolds number have been studied by changing the flow viscosity.

Keywords: The best mesh in staggered configuration, Flow quantities, staggered tube bundles.

HIGHLIGHTS:

1- Laminar and Turbulent flow in staggered tube bundles.

2- The behaviour of flow in staggered tube banks.

3- A comparison of three different meshes with a very fine grid.

Corresponding author: Nabeel Abed

Author affiliation: Assistant lecturer

Email: nabilkhalil90@gmail.com

Mobile: +9647803021102

Mechanical Technical Department, Institute of Anbar, Middle Technical University, Iraq.

International Journal of Scientific & Engineering Research Volume 9, Issue 7, July-2018
ISSN 2229-5518

1-Introduction

A bank of tubes is a sub-component of shell-and-tube heat exchanger which commonly used in many engineering applications such as cooling and heating systems and nuclear power plants in which the nuclear rod of fuel has been cooled by water passing around them. The tubes in heat exchanger can be arranged usually either in in-line configuration or in staggered configuration described by the longitudinal, transverse and diagonal pitches. Typically, two unmixed flows are involved in the tube banks where one of them moves inside the tubes whereas the other flows around the tubes see figure 1. The analysis of heat transfer and pressure drop in the tube bundles is highly important in terms of the design of heat exchangers. So as to obtain a reliable calculation of heat transfer, the flow around the tubes must be properly resolved using computational approach.

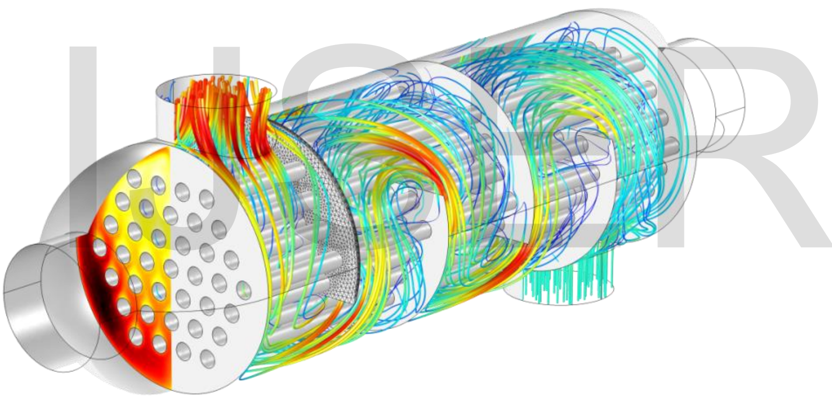


Figure 1: CFD simulation for shell and tube heat exchanger [(1)].

Indeed, in every single CFD work, the very important point that needs to be considered is the grid generation. That is due to a good generating of grid leads to provide some factors which in turn give a very good agreement with the experimental work. These factors are related to the numerical solution, not to the flow solver which are CPU time, convergence rate and the accuracy of the numerical solution.

(1) See: http://www.3dcadworld.com/wp-content/uploads/2013/11/shell-tube.png.

(Acceded at 11/11/2017).

Consequently, quicker convergence and better accuracy can be obtained by generating a very fine grid. Therefore, it is important to cast a shadow on advantages and disadvantages of meshes types.

In order to resolve the main features of the analysed problem properly, the mesh being used should be fine enough. The resolution of the mesh is dependent on two things; firstly, the boundary mesh from which the starting island secondly, the parameters which control the inner mesh generation. The changing in the size from one cell (or face) to the adjacent one in the high-quality mesh should be reduced because the poor computational results will be achieved if the difference in the size of adjacent cells (or faces) is large. This is due to the differential equations which are solved assume that the cells shrink or grow without difficulties (smoothly).

Liang and Papadakis, (2005) studied the effect of turbulent flow using LES on both inline and staggered arrangements for different Reynolds numbers. They concluded that the staggering results provided better performance than those of inline results. Bahaidarah et al. (2005), Jeng and Tzeng (2007) presented a numerical study using both configurations (inline and staggered) and they found that the staggered arrangement showed better performance for heat transfer than those of inline tube banks. In (2011) a paper has been published by Salinas-Vázquez et al. of a CFD study using LES approach to investigate the flow in 3D, they found that the staggered arrangement provides an excellent agreement with the experimental work. Therefore, the staggered configuration has been selected in this work.

In the present study, the laminar cross flow over a staggered arrangement has to be investigated using Star CCM+ in three different meshes namely; block structured quadrilateral cells, an unstructured mesh made of quadrilateral cells (PAVE) and mesh (TETRA) which is made of triangular elements.

2- Some advantages and disadvantages of structured grids and unstructured grids

Block-structured mesh:

In this mesh, the domain has been divided into different blocks, it seems to be unstructured globally when presented as a collection of blocks; however, it is locally structured in an individual block point of view. That allows to the most adequate grid arrangement to be used in each region to construct the whole domain individually in the physical geometry and also this would provide the user with more freedom in terms of building the mesh, it is widely used in the complex geometries which have consisted of several geometrical sub-components like the internal combustion engine pent-roof cylinder and inlet port geometry, see figure 3 (Versteeg and Malalasekera, 1995).

The solver of block-structured flow usually requires the minimum amount of memory for the size of mesh and can be executed quicker due to the optimization of this mesh. In addition to that, the post processing of the results on this mesh is usually very easier task due to the planes of this grid has been arranged logically. Moreover, it gives better convergence and higher resolutions.

On the contrary, this mesh bring disadvantage as well, the main one is this grid requires time and experience to place the optimal block structure for an entire mode.

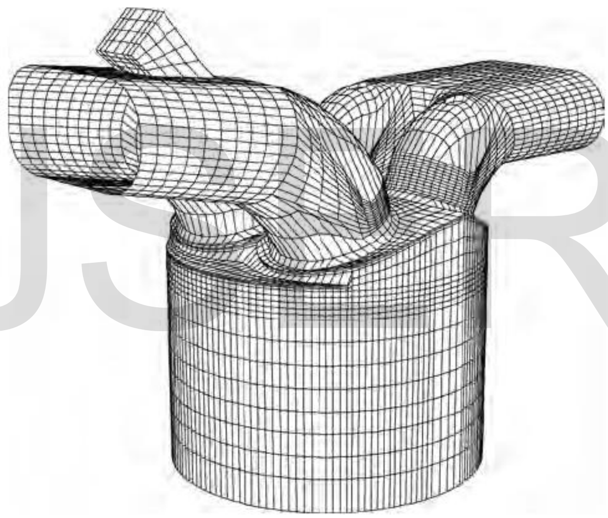


Figure 3: Block-structured mesh for engine pent-roof cylinder and inlet port geometry, from (Versteeg and Malalasekera, 1995).

Unstructured grid:

In this type of meshes, the elements are collected arbitrarily in order to fill the geometry domain, therefore, because the configuration of elements in the grid are not discernible that it is called unstructured, (Liseikin, 2007).

Advantages:

More control of changing the aspect ratios can be allowed in the unstructured grid and thus can be refined locally. Also, the solver of algebraic equations in this grid is in general slower than those of structured grid. In addition to that, it is easy to use; thus, it needs just a few time and effort. Moreover, it is suitable for inexperienced users, a valid mesh in most cases is generated due to a little effort. It enables the solution of very large problems in short time. The times of grid generation can be in general measured in minutes or hours. Connectivity information is more costly, it is however limited to immediate neighbours, so the order of discretisation is low, 1st or 2nd order, in unstructured finite volume.

On the other hand, many disadvantages are involved in this type of mesh which are; the shortage of user control when designing the mesh. Moreover, the involvement of any user has been limited to the mesh boundaries with a mesher automatically filling the interior. In addition to that, it is limited to be large isotropic due to the elements of the grid are not be stretched or twisted well, thus, all the grid elements have approximately the same shape and size. Therefore, it is a big problem when attempting to refine the grid in a local area, often the entire grid must be done much finer so as to obtain the point densities desired locally. Also, for the same mesh, the solver of this grid generally needs larger memory and has longer running times than this of structured grid. The Post-processing of results presentation has required powerful tools for interpolating the results, (Versteeg and Malalasekera, 1995).

3-Numerical Procedure

The laminar flow over a staggered tube banks is numerically investigated using Star CCM+ on three different meshes and the computational domain is highlighted as in figure 2, which would provide symmetries in the flow.

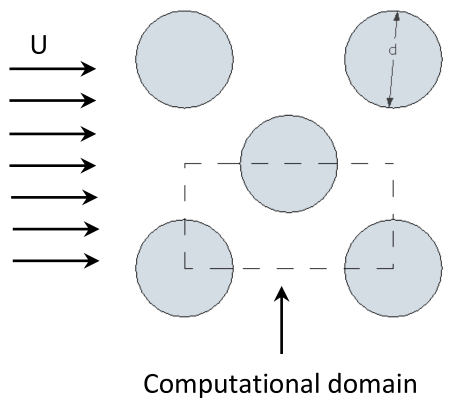


Figure 2: The computational domain of the present work.

Apply the periodic conditions (with a prescribed pressure drop) between the east and west boundaries whereas symmetry conditions on the top and bottom boundaries. Then, run the first mesh which called block-structured quadrilateral cells with a resolution of 540 cells in total with 72 points around one cylinder, and take the contours of velocity and pressure and also the grid itself in order to make a comparison between the performances of them. After that, run the second one which called an unstructured mesh made of quadrilateral cells (PAVE). The total number of cells is 433 with 64 points around one cylinder, and also takes the contours of velocity and pressure and the grid itself. Then, the same procedure can be done with the third one which is (TETRA) is made of triangular elements, with a total number of 490 cells with 64 points around one cylinder. In order to know which one can be considered is very close or at least try to be near from the converge solution, the converge solution of very fine grid is provided and can be easily made a comparison between the results of these three meshes (velocity, pressure, etc.) and with those of the very fine grid and the weakness and strengthens of each mesh should be examined.

4-Results Presentation and Discussion

As can be seen in the figure 5, the fine mesh, which has mesh dependent solution, indicates the reference point to others in the contours of velocity distribution, thus, it has the largest level of accuracy and the smallest percentage in terms of error owing to the large number of cells performed around the tubes and also the organization of these cells is done well without any deformity in the mesh. Moreover, the fine mesh is a hybrid of an unstructured grid where quadrilateral cells are employed near walls in order to give better results in terms of viscose influences in the boundary layer; however, triangular structure mesh is applied in the rest locations in the domain in order to use the resources in an efficient way. The maximum velocity, as shown, is at the centre region of the flow whereas the minimum velocity is at the near wall regions.

The results of block-structured mesh are more close to those of the fine mesh than the results of the others behind many reasons; firstly, the generation of this block-structured mesh is easy due to the easy discretization of the equation we deal with, and also curved boundaries have to be easily accommodated, thus multi blocks might

International Journal of Scientific & Engineering Research Volume 9, Issue 7, July-2018
ISSN 2229-5518

be handled with desired degree of fineness in mesh. Secondly, the refining in the zones in which the geometry needs to be captured more precisely. Thirdly, the cell number around the cylinder is 540 which is larger than those of PAVE and TETRA meshes, thus the deformation in this mesh is small comparing with the deformation of other meshes and that afforded it to give better solution.

In the pressure contours in figure 6, it is obvious that the block-structured grid is still the first one closer to the fine grid in terms of higher accurate and lesser errors whereas the unstructured grid is still the worst. It can be clearly seen that the lowest values of pressure are present at the bottom and the top of the cylinders owing to the largest values of velocity at these points. On the other hand, the largest values of pressure are at the stagnation points in which the velocity direction is almost normal to the cylinder. This is caused by fluid acceleration due to the reduction in flow area. In the staggered configuration, there is further decreasing in the pressure owing to the acceleration and separation from the tube walls. Consequently, the losses of friction pressure are larger in the staggered tube bank configuration. As expected, the pressure is shown to gradually decrease and increase as the flow moves around the tubes. The reduction in the flow area caused the pressure drop as the flow has moved towards the minimum gap. Pressure recovery occurs as the flow separates from the tube just after the minimum gap and expands to re-attach to the next tube. There is a net pressure drop across each tube due to friction.

If the rule of midpoint integral approximation, linear interpolation and central differences has been used to discretize the governing equation then higher accuracy would be obtained than those due to the usage of the quadrilaterals in 2D using tetrahedral mesh. The reason behind that is a high level of errors which have been made at opposite cell faces when discretizing diffusion terms cancel partially (if cell faces are parallel and of equal area, they cancel completely) on quadrilateral CVs. Therefore, more sophisticated interpolation and difference approximations need to be employed particularly near wall regions in order to obtain the same accuracy and that tells us that the topology of the cell is very important.

In addition, the accuracy can also be improved for convection terms if the collection of the grid lines are able to closely follow the flow streamlines; however, this definitely would not be obtained by using triangles and/or tetrahedral meshes, (Ferziger and Peric, 2002).

The figure 7 represents the relation between the velocity in the stream-wise direction versus the y position in the special domain. It is obvious that the velocity has decreased initially until it reached to the maximum negative value and then started rising gradually until it reached to the maximum positive value. This is due to the separation bubble, the flow after that has reattached and increased gradually until reaching to the next tube and because of the impingement, the flow decreased to the zero.

The distance between two positions in which the velocity fluctuates from negative to positive value is known as the recirculation length which indicates the length of the separation bubble in the wake region behind the tube.

The different results have been obtained by different meshes depending on their ability to capture the flow physics at these positions when the flow has passes over the tubes.

As can be seen that the TETRA provided undesired results and gave not good agreement with the results of the very fine mesh and this is due to the not equilateral triangular cells employed around the boundary regions which are responsible for unacceptable physical values at some points and hence this does not afford it an opportunity to capture the changes in the near wall region. Additionally, the PAVE provided generally under-estimation results and also not in a good agreement with those of the very fine mesh. However, the block-structured grid showed acceptable results and it seems to be a better grid comparing with others, and as it is mentioned before this type of grid has an ability to provide better convergence and higher resolutions.

As can be seen from figure 8, the profiles of laminar flow that have been captured by three different meshes are in a better agreement with that captured by the very fine mesh than the previous measuring probe along (y=0). That might be because the skewness in this region was controlled in which the boundary is periodic and the variables are almost uniform. Another distinguished feature can be noticed in figure 8 which is at the end of the curve (near wall region), the values of U- velocity at three different meshes did not reach to the zero while it did at the very fine mesh. Therefore, if the boundary layer has needed to be resolved, the TETRA mesh is not desirable near to the wall because the first point (the size of cell should be very small) must be very near to the wall in order to solve the viscous effect whereas the large size of grids might be employed in the parallel direction to the wall. This requirement is leading to long thin TETRA, making problems in the diffusion approximation. Because of that reason, the fine grid has quadrilateral cells near the walls while a TETRA mesh has been generated automatically in the remaining part of the domain. Unfortunately, that occurred not only with TETRA mesh but also with PAVE and Block-structured meshes as well. Thus, more reduction in cell size needs to be in order to improve the behaviour of the meshes.

An addition error source can be noted is that the central differences that have involved in the integration of control surface element in the unstructured grid are just second-order accurate. This error might increase with the increasing the aspect ratio and skewness, see figure 9 which shows the skewness in both types of meshes; TETRA and PAVE.

In the table 1, there is the comparison between the values of U-bulk, Re number , pressure drop coefficient (Kp), Recirculation Length, and the error for each grid in terms of velocity. In general, the results obtained by block-structured grid provided good agreement with those of the very fine grid where the error of velocity was 1.43E-2 % while the errors of other grids (TETRA and PAVE) are almost more than two times this of a block-structured grid. However, the recirculation length in TETRA mesh is very closer to that of the very fine mesh than those of other two meshes.

Turbulent work discussion

The most work in this section is focused on just the block-structure grid beside comparing with the very fine one because the block-structured grid was the best choice in the preceding comparisons; therefore, there is no point in repeating here.

As can be seen in figure 10, in the mainstream the flow is very high and follows a very tortuous path using a standard k-ε model. When the flow has past the cylinder, which is placed at a no-slip condition at the wall, it is clear that the fluid flow has decelerated close to the surface of the cylinder and caused in generating a very thin layer which called boundary layer owing to the viscous effects near-wall region. The fluid flow has been attached to the surface of the cylinder until the creation of wake at the rear part of the cylinder, where some flow has followed backward opposition to the mainstream. It is clear that the largest velocity has occurred at $\theta = 90^{o}$. However,

close to θ=180º, the velocity has been at the smallest or can be said zero, in this region the recirculation happens and in the cylinder front, the flow has re-attached. Another feature can be noticed from this figure is that in the block-structured grid, which is a better grid comparing with PAVE and TETRA, was not able to capture a huge vectors of velocity in the re-circulation region due to its coarse and the mesh generation errors associated and the question is who the flow will be captured by other grids in this regions?
Definitely more worst. However, just the fine mesh has an ability to capture very smooth vectors which give an idea that this mesh has been refined well and provides; therefore, very accurate results.

As shown in figure 11 (a), the pressure distribution provided smooth behaviour which is more smaller than of those in laminar flow due the high velocity of flow expect in the stagnation points at the corners in red colour. In spite of that, the pressure affected regions are smaller than those in laminar flow. However, in figure 11 (b) the velocity distribution gave further increase in the main flow due to higher velocity involved and; consequently, led to the boundary layer becomes more thinner because the flow now is dominated by the inertia force where the effect of viscosity force is quite small. In terms of mesh quality, the very fine mesh shows high quality as shown in the figure 12, and; therefore, the results presented in high level of accuracy because it has the largest level of accuracy and the smallest percentage in terms of error owing to the large number of cells performed around the tubes and also the organization of these cells is done well without any deformity in the mesh. Moreover, the fine mesh is a hybrid of an unstructured grid where quadrilateral cells are employed near walls in order to give better results in terms of viscose influences in the boundary layer.

Figure 13 represents the relation between the pressure coefficient (K_p) versus the Reynolds number. It is obvious that the pressure coefficient is proportional inversely with the Reynolds number in which when the last has decreased owing to increasing the viscosity (multiplied by 2), the pressure coefficient has increased approximately twice. By decreasing Reynolds number again due to increasing the viscosity (multiplied by 4), the pressure coefficient jumps considerably to more than the previous one which leads to increase the pressure drop in the tube bundles according to the relation between them which is ($\Delta P = (0.5)\ K_p\ U_b^2\ \rho$). In terms of meshes issues, it is expected that there is no a huge difference would be among the four current meshes due to the decreasing of the flow velocity (increasing viscosity) and the flow is dominated by the viscous force. Therefore, the opportunity for capturing the flow physics is now larger even if the coarse meshes are used and that led to a high accuracy and very small difference in results of the block and very fine meshes.

The wake flow behind the tube can be characterized by several non-dimensional quantities such as recirculation length L_R/D and the separation angle θs, this interpretation is the same of that occurred behind a single cylinder. As can be seen in figure 14, the recirculation length is directly proportional with Reynolds number. When the Reynolds number has decreased due to increasing the viscosity (multiplied by 2) the recirculation length drop significantly as shown and with further reducing the Reynolds number (where viscosity is multiplied by 4), the recirculation length becomes very small and can be said it has vanish. That is caused due to the fact that the flow is dominated by the viscous force overwhelmingly where the inertia force becomes very small. Therefore, the flow layers become highly smooth and adjacent layers slide past one another without mixing and no disruption as well. In terms of the grid accuracy, it is obvious that the block-structured grid provided good agreement with the very fine mesh except a little overproduction at the case (where viscosity multiplied by 2) and then returned to the same value in agreement with a very fine mesh.

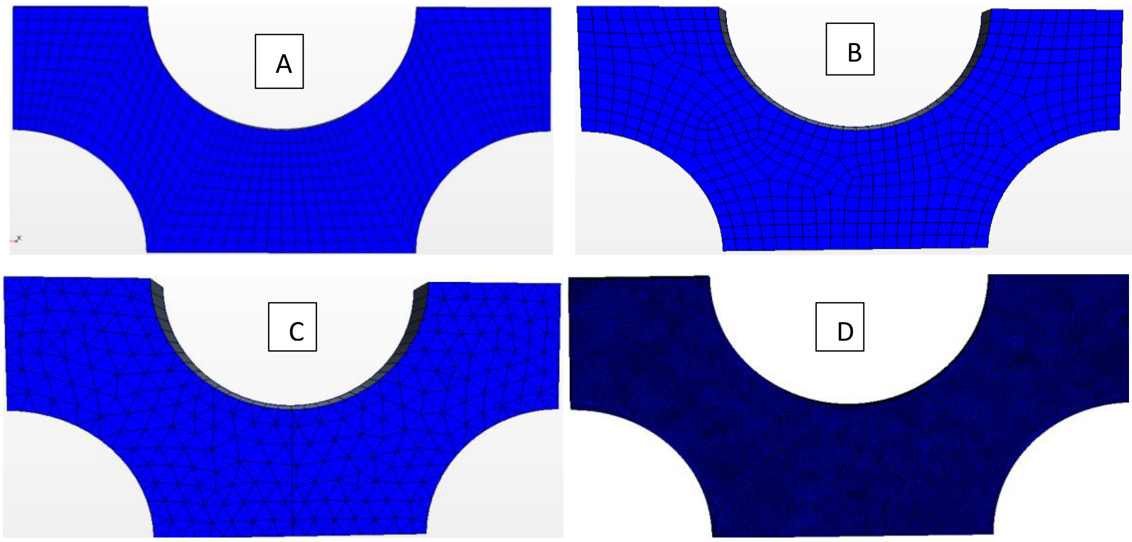


Figure 4: The four different meshes, A: block-structured, B: PAVE, C: TETRA and D: the fine mesh.

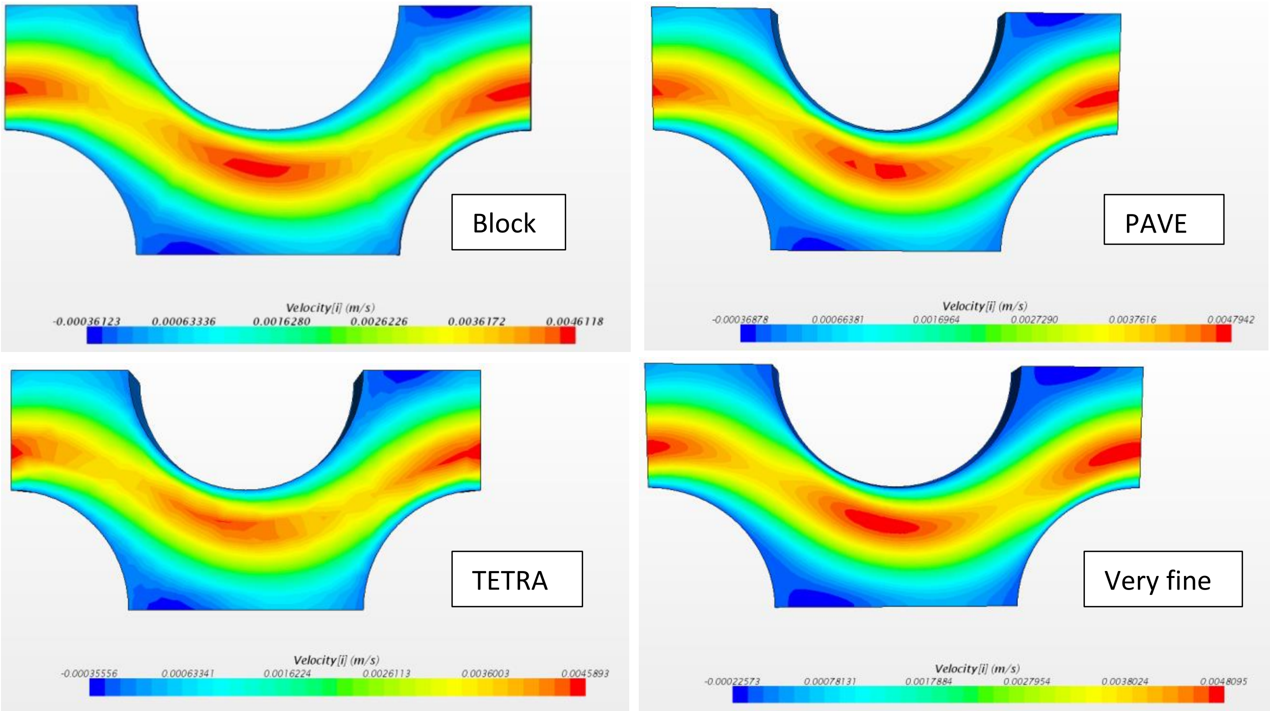


Figure 5: The velocity contours for four different meshes.

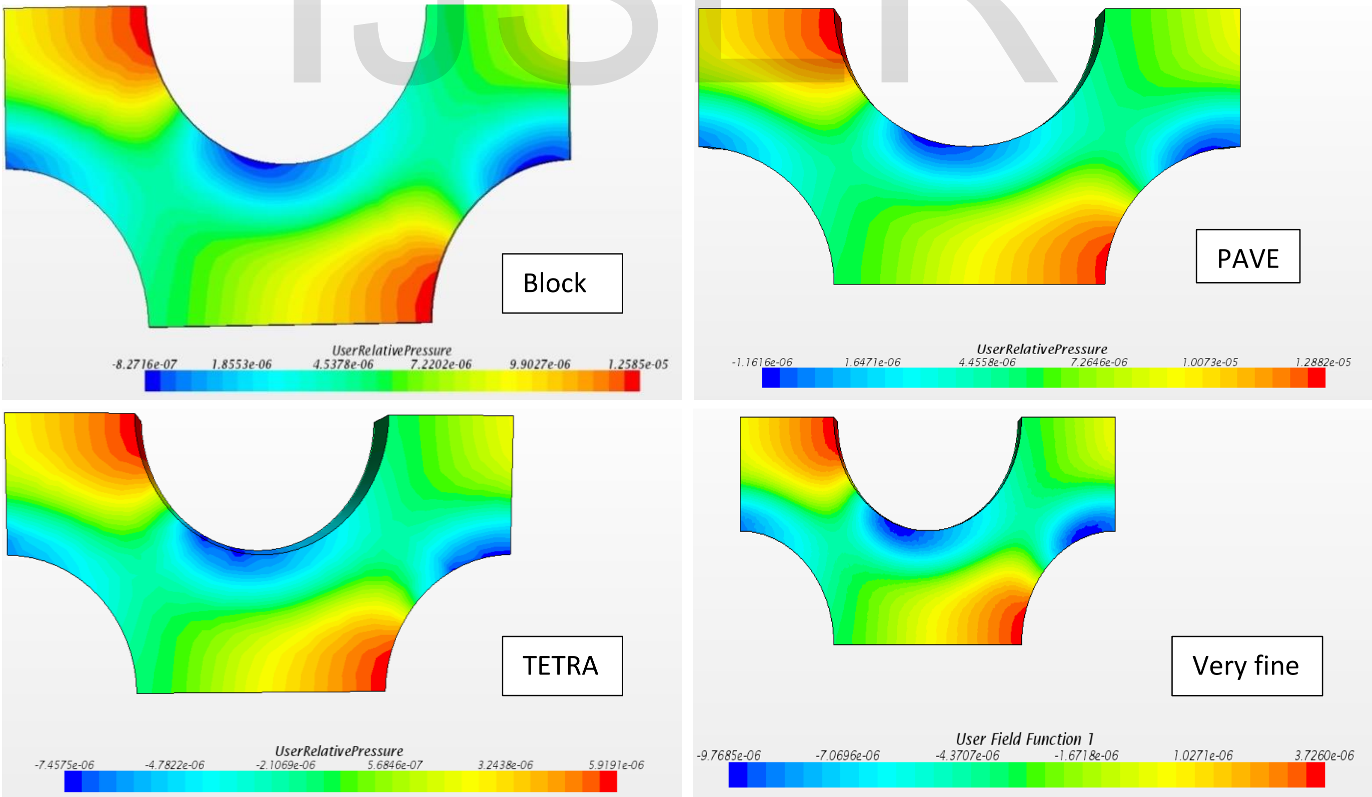


Figure 6: The contours of pressure for four different meshes.

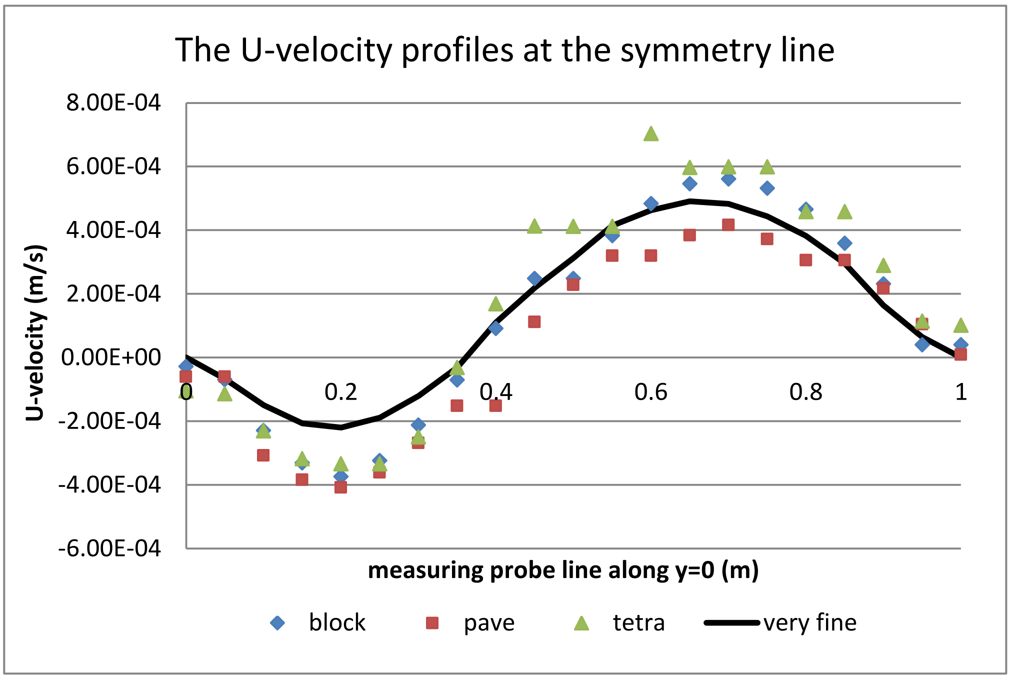


Figure 7: The U-velocity profile for different grids with the fine mesh at the symmetry line (y=0).

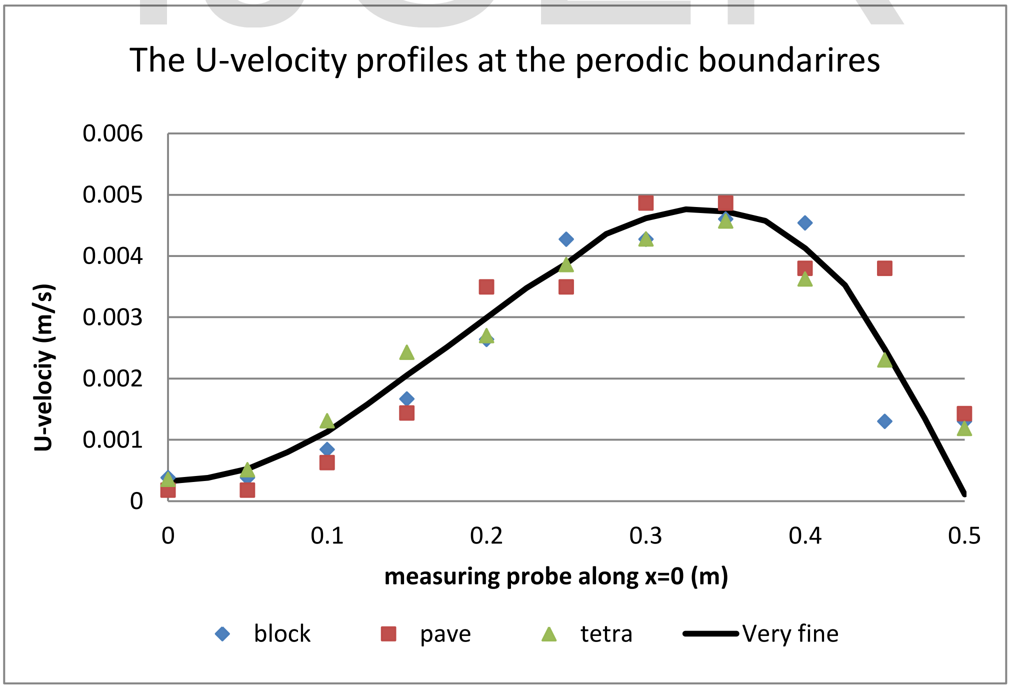
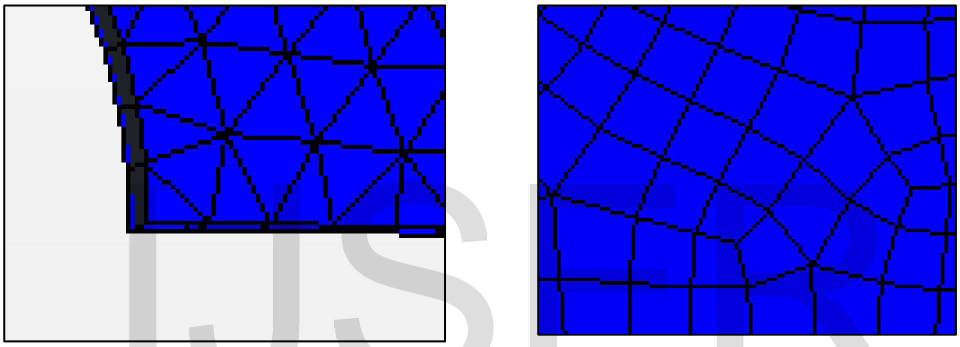


Figure 8: The U-velocity profile for different meshes along the periodic boundary (x=0).

Table 1: The values of U-bulk, Re number, pressure drop coefficient (kp), Recirculation Length, and the error for each grid in terms of velocity.

Mesh	U-bulk (m/s)	Re number	Kp	Recir. length(m)	Error (%)
BLOCK	2.75E-03	177.4094	2.24	0.3717	1.43E-02
PAVE	2.81E-03	180.8985	2.15	0.0571	3.43E-02
TETRA	2.80E-03	180.4204	2.16	0.358	3.15E-02
Very Fine	2.72E-03	174.9077	2.30	0.361	



(a) (b)

Figure 9: The skewness in the TETRA (a) and PAVE (b).

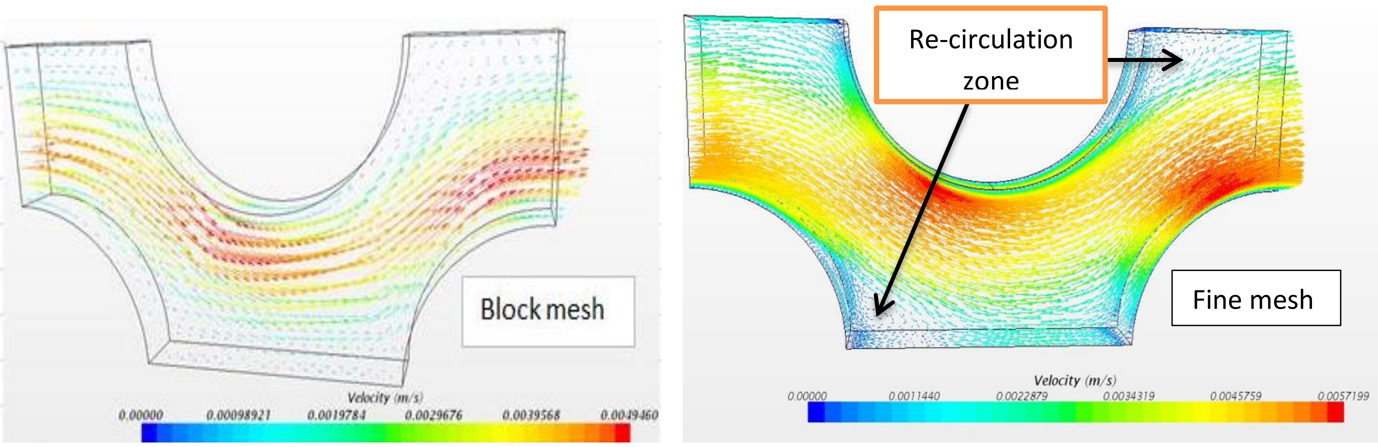
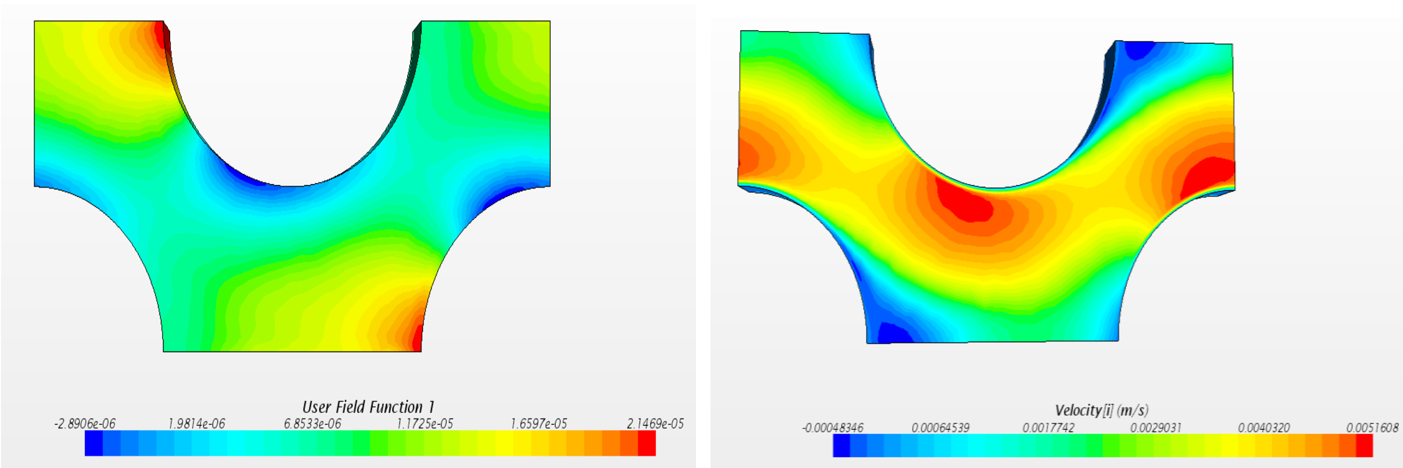


Figure 10: Velocity vectors in turbulent using the fine mesh and block mesh.



(a) (b)

Figure 11: the contours of pressure (a) and velocity (b) for turbulent flow using very fine mesh.

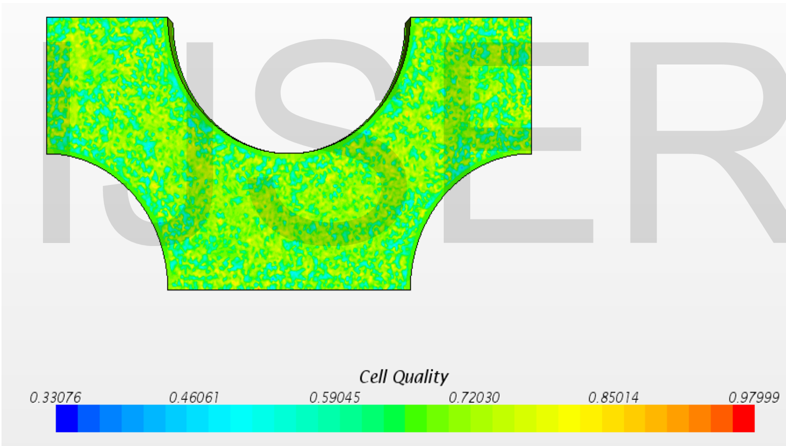


Figure12: the quality of the very fine mesh.

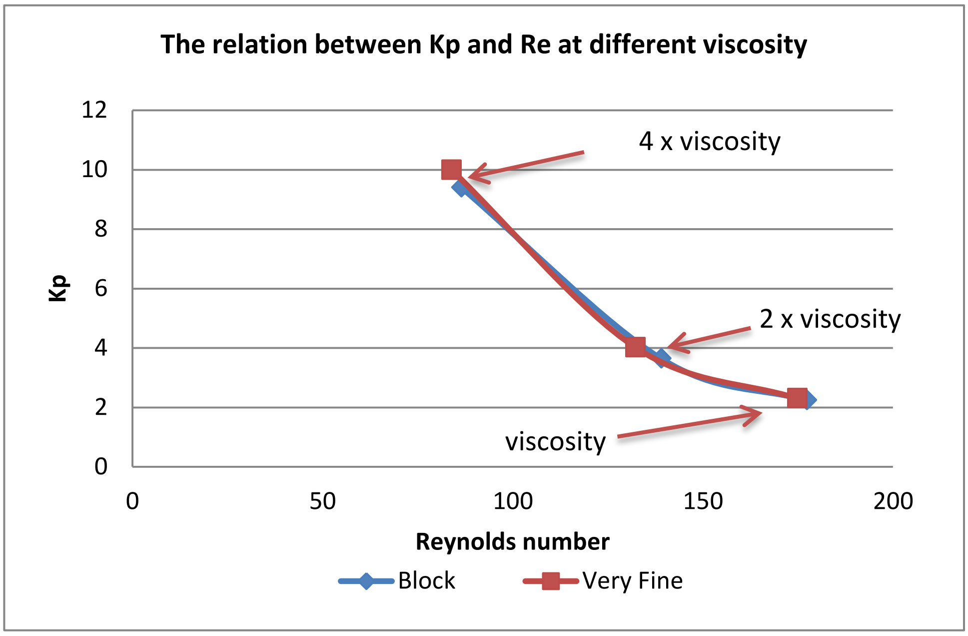


Figure 13: The relation between pressure coefficient and Reynolds number with changing the viscosity.

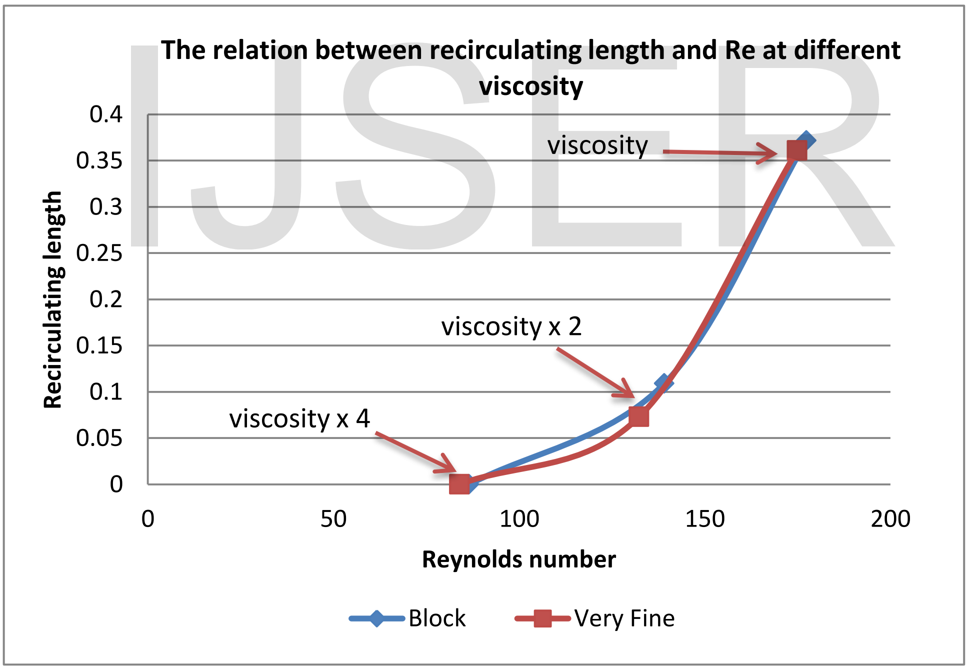


Figure 14: The relation between the re-circulating length versus Reynolds number with changing the viscosity.

5-Conclusion

The effect of changing the mesh in staggered tube bundles was numerically investigated using STAR CCM+ as a flow solver. Three meshes were applied to show the best agreement comparing with the very fine grid. The three different meshes are: block structured quadrilateral cells, unstructured grid quadrilateral cells (PAVE) and TETRA mesh made of triangular cells. The block mesh provided the system with several advantages; the best results of the problems have been guaranteed, the running for the additional analysis was reduced, and the predictive capabilities are improved. Therefore, the block mesh has been selected as the best mesh and very suitable grid for the flow in staggered tube banks. However; in turbulent flow, the standard k- ε model has been employed. The relationship of changing the pressure drop coefficient with Reynolds number was indicated by changing the flow viscosity. Moreover, the relationship between the recirculation length and Reynolds number was also investigated by changing the viscosity of flow.

6-Reference

Bahaidarah, H., Anad, N. K., & Chen, H. C. (2005). A Numerical Study of Fluid Flow and Heat Transfer over a Bank of Flat Tubes. Numerical Heat Transfer: Part A: Applications. 48, 359-385.

FERZIGER, J. H., & PERIĆ, M. (2002). Computational methods for fluid dynamics. Berlin, Springer.

Jeng, T.-M., & Tzeng, S.-C. (2007). Pressure drop and heat transfer of square pin-fin arrays in in-line and staggered arrangements. International Journal of Heat and Mass Transfer. 50, 2364-2375.

Liang, C. and Papadakis, G. (2005). Study of the Effect of Flow Pulsation on the Flow Field and Heat Transfer Over an Inline Cylinder Array Using LES.

LISEĬKIN, V. D. (2007). *A computational differential geometry approach to grid generation*. Springer E-Books. Berlin, Springer. http://public.eblib.com/choice/publicfullrecord.aspx?p=304623.

Salinas-Vázquez, M., M.A. de la Lama, W. Vicente, and E. Martínez. 2011. "Large Eddy Simulation of a flow through circular tube bundle". *Applied Mathematical Modelling.* 35 (9): 4393-4406.

VERSTEEG, H. K., & MALALASEKERA, W. (1995). An introduction to computational fluid dynamics: the finite volume method. Harlow, Essex, England, New York.