INNOVATIVE MECHANICAL ENGINEERING **ISSN 2812-9229 (Online)**
University of Niš, Faculty of Mechanical Engineering
VOL. 2, NO 2, 2023, PP. 14 - 25

Original scientific paper *

CFD SIMULATION OF MICROPOLAR FLUID FLOW IN HORIZONTAL CHANNEL

Miloš Kocić[1], Živojin Stamenković[1], Jelena Petrović[1]

[1]Faculty of Mechanical Engineering, University of Niš, Serbia

Abstract. *Computational fluid dynamics (CFD) is a numerical analysis of the problem of fluid flow, heat transfer and determination of all fluid parameters in the flow field. However, none of the modern available software (Fluent, CFX, Open FOAM etc.) has a micropolar type of fluid defined in the palette of available fluids, and nowadays there is a need to find a way to analyze the flow of such fluids through computational fluid dynamics. Therefore, the paper will present a CFD model for the analysis of micropolar fluid flow in a horizontal channel. The validation of the results will be obtained by comparing the obtained results with the analytical model, and the obtained results will be analyzed through the graphs of the influence of the characteristic parameters on the physical characteristics of the flow, as well as through the visualization of the results directly from the ANSYS CFX software.*

Key words: *CFD, MHD, Micropolar fluid, Fluid flow*

1. INTRODUCTION

The development of computers and IT technologies, and then software for numerical simulations, enabled another new field of fluid flow and heat transfer analysis. Computational fluid dynamics (CFD) is a numerical analysis of the problem of fluid flow, heat transfer and determination of all fluid parameters in the flow field. Parallel to the expansion of computers, CFD has been accepted as an important tool for performing fundamental research and flow calculations of interest to practice. CFD solvers contain a complex set of algorithms that are used for modeling and simulating the flow of liquids and gases, analysis of heat transfer and all other quantities that are modeled in the analyzed problem. Many technological advances in aerospace, car industry and astronautics would not be possible without CFD. Applications of CFD in the study of airfoils in aviation, then the analysis of the stopping track of a car or the track of braking, the analysis of flow and thermal processes during the design of jet engines, as well as the cooling of electronic components by air flow, are only some of the practical problems where numerical simulations have proven to be very useful [1, 2].

*Received: December 05, 2023 / Accepted December 21, 2023.**
Corresponding author: Miloš Kocić
Faculty of Mechanical Engineering University of Niš, Serbia
E-mail: milos.kocic@masfak.ni.ac.rs

CFD simulations have a significant advantage over empirical experiments. Conventional experiments allow the analysis of data only from a limited number of places in the system (where sensors and meters are located). CFD simulations, on the other hand, allow researchers to explore any location in a region of interest and interpret its performance using multiple thermal and flow parameters. In addition, using classical experiments and tests to obtain the necessary engineering data for design is much more costly. An additional advantage is the fact that many simulations can be performed in a much shorter time than by conducting laboratory tests [3].

The basis of CFD is built on the Navier-Stokes equations and turbulent flow models, i.e., a set of partial differential equations that describe fluid flow [4]. With the help of CFD, the analysis area is divided into a large number of cells or control volumes. In each of these cells, the Navier-Stokes partial differential equations can be written as algebraic equations relating the velocity, temperature, pressure, and other variables to the values in the adjacent cells. These equations are then solved numerically, giving a complete picture of the flow whose accuracy depends on the "fineness" of the network, i.e., dependent on the number of cells. The resulting set of equations can then be solved iteratively, giving a complete picture of the flow over the entire domain. By solving the fundamental equations that mathematically describe fluid flow processes, CFD provides information on important flow characteristics, such as pressure drop, velocity or temperature profile. It can be said that CFD analysis complements traditional testing and experimental research, providing additional insight and confidence in developed projects. This results in better product design, as well as lower risk and shorter time to arrive at a finished product or process.

These techniques date back to the early 1970s, and the first commercial CFD software became available in the early 1980s [5]. CFD has come a long way since then, and now there are very few geometrically complex problems that cannot be accurately represented. In the meantime, models have been developed for physical phenomena such as turbulence, multiphase flow, various chemical reactions, and heat transfer, and the usability of the software has increased significantly with the increase in computer capacities. However, even today's modern software (Fluent, CFX, Open FOAM etc.) do not have developed models for the flow of micropolar fluids.

Flows of micropolar fluids are of great practical importance today. Their application can be found both in various branches of industry and in medicine. Some examples of micropolar fluids are human and animal blood, polymer fluids, liquid crystals, various types of lubricants with additives, etc. Micropolar fluids are fluids with microstructure. They belong to a class of fluids with nonsymmetric stress tensor that we shall call polar fluids, and include, as a special case, the well-established Navier-Stokes model of classical fluids. Physically, micropolar fluids may represent fluids consisting of rigid, randomly oriented (or spherical) particles suspended in a viscous medium, where the deformation of fluid particles is ignored [6]. One of the best-established theories of fluids with microstructure is the micropolar fluid theory by A. C. Eringen [7].

In the last five decades, research on the flow of micropolar fluids has attracted a lot of attention. Recently published works on this topic [8, 9] show that the importance of researching these types of flows is still relevant. However, the flow analysis of these types of fluids is exclusively related to the solution of complex flow equations, either analytically or using one of the numerical methods, while the possibility of using computational fluid dynamics (CFD) is practically impossible due to the lack of an adequate model for micropolar fluid in modern software.

Considering everything previously said, the aim of this work is to show that in the ANSYS CFX software, a detailed analysis of simpler types of micropolar fluid flow can be performed, using a combination of already offered flow models in the software palette itself. All this is done with the desire that in the near future a specific model for the study of these types of fluids may be developed.

2. Model of considered problem

2.1 Physical and mathematical model

The model of laminar and developed EMHD flow of micropolar fluid in the channel will be considered. With the assumption that the flow is mainly in the direction of the x axis, and that the Reynolds magnetic number is significantly less than one, the observed problem is simplified. During fluid flow, the upper and lower plates are kept at constant temperatures T_{w1} and T_{w2}, respectively, and these plates are assumed to be electrically conductive. During the flow, an external magnetic field of intensity B in the direction of the y axis and an external electric field of intensity E in the direction of the z axis, act on the micropolar fluid. The previously defined physical model is given in Figure 1.

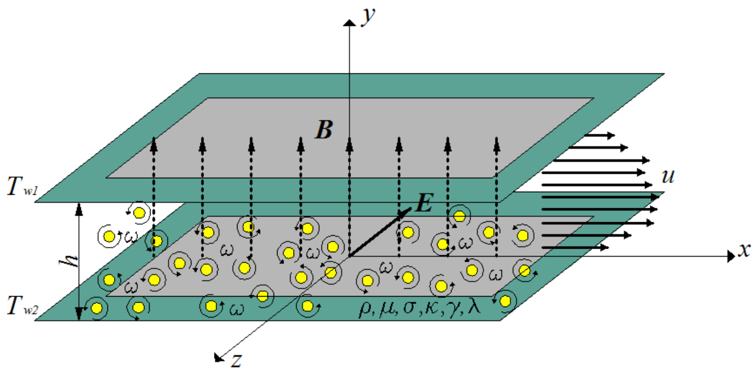

Fig. 1 Physical model of EMHD flow of micropolar fluid

The velocity, magnetic and electric field vectors are given by the following expressions:

$$\mathbf{v} = u\vec{\imath},\ \ \mathbf{B} = B\vec{\jmath},\ \ \mathbf{E} = E\vec{k}, \tag{1}$$

while the system of equations for micropolar fluid flow is given by the following expressions:

$$(\mu + \lambda)\frac{d^2u}{dy} + \lambda\frac{d\omega}{dy} - \sigma B^2 u - \sigma EB - \frac{dp}{dx} = 0 \tag{2}$$

$$\gamma\frac{d^2\omega}{dy^2} - \lambda\frac{du}{dy} - 2\lambda\omega = 0 \tag{3}$$

$$\kappa\frac{d^2T}{dy^2} + (\mu + \lambda)\left(\frac{du}{dy}\right)^2 + \sigma(E + uB)^2 = 0 \tag{4}$$

To solve the previous system of equations (2) - (4), it is necessary to introduce appropriate boundary conditions:

$$\begin{cases} u = 0, \omega = 0, T = T_{w2} \ \ for \ \ y = 0, \\ u = 0, \omega = 0, T = T_{w1} \ \ for \ \ y = h. \end{cases} \quad (5)$$

The considered mathematical model of micropolar fluid flow has already been solved and presented in previous manuscript [10], so it will be used here only as a control model for comparing the results obtained through numerical simulations (CFD).

2.2 CFD model in ANSYS CFX

One of the most commonly used software packages for CFD simulations is ANSYS [11]. As part of this software package, ICEM-CFD will be used first to form the geometry and network of the flow space, then from the ANSYS software package CFX-Pre, where all pre-processor descriptions of the flow and heat transfer problems are performed, after that CFX-Solver, which is used to solve the equations, and finally CFX-Post or the post-processing part of the software package where the analysis and display of the obtained results is performed.

When solving the fundamental equations that describe fluid flow, there are several different methods used in CFD codes today. With ANSYS CFX-Solver, the most common method is the method of finite volumes. With this method, the domain in which the flow is considered is divided into small subregions, called control volumes. The equations are discretized and solved iteratively for each control volume. As a result, the approximate value of each variable can be determined in an arbitrary control volume of the domain under consideration. In this way, a complete current picture of the considered fluid flow problem is obtained.

The process of modeling and analysis of micropolar fluid flow in the ANSYS software package, and based on the previous mentioned parts of that package, can be presented in four basic phases:

- first stage - defining the geometry and network for the considered problem,
- second phase - defining the basic characteristics of the micropolar fluid, as well as the boundary and initial conditions of the considered problem,
- the third stage - solving the equations of the problem,
- fourth phase - processing and analysis of the obtained results.

When analyzing the flow of a micropolar fluid in a channel using the ANSYS software package, the first challenge was to define a micropolar fluid because there is no micropolar fluid in the database of available fluids. As the micropolar fluid was previously defined as a fluid with a microstructure that contains randomly oriented (mostly spherical) particles dissolved in a viscous liquid, while the deformation of these particles is ignored, the starting idea was to observe the micropolar fluid in ANSYS as a flow of solid particles in viscous fluid, through the option that this software offers called "particle transport solid". This made it possible to define additional particles and define additional viscosity that characterizes the flow of micropolar fluids λ.

The next step was to define the geometric model and mesh [12]. The channel model has dimensions of 1000 mm x 20 mm x 50 mm in *x*, *y* and *z* axis directions, respectively, and is given in Figure 2.

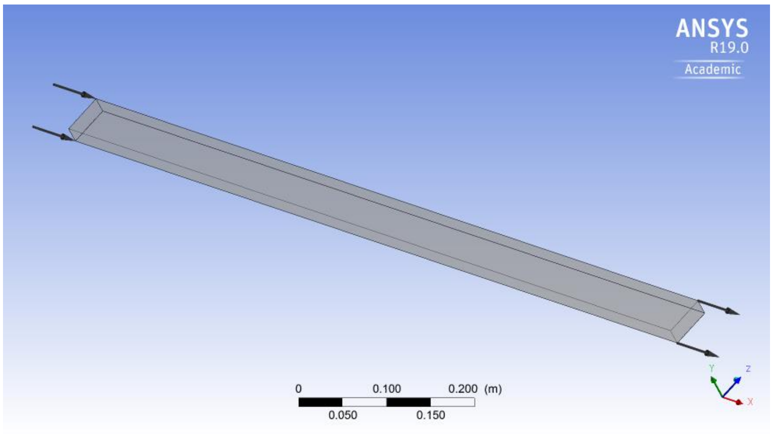

Fig. 2 Channel model from ICEM-CFD

When defining the model, it is important to emphasize that it is a laminar flow, and that the flow of a micropolar fluid is the result of a constant drop in pressure along the axis of the flow, *x* - axis. Accordingly, the boundary conditions at the inlet and outlet of the flow domain are defined by the flow at the inlet and the constant pressure at the outlet of the flow domain. What was also important when defining the boundary values at the entrance was defining the mass flow of elementary particles, as well as their number and size. As for the other boundary conditions, there is no slip on the upper and lower plates, i.e., the speed is equal to zero, while on the walls perpendicular to the *z* axis the so-called symmetric boundary conditions, i.e., the problem is considered as a two dimensional.

Regarding the definition of the mesh, it is very important to determine a sufficient density of the mesh that shows agreement with the laminar velocity profile when the fluid flows between the parallel plates [13]. This task is called "mesh independence test", test of independence of results from increasing network density. In addition, it is very important to take care when forming the flow domain, and the mesh, that the domain in the direction of the flow is at least ten times larger than the transverse coordinate (distance between parallel plates). Taking the previous criteria, as well as the flow conditions, a network is formed that has 150 nodes uniformly distributed in the *x* direction, or direction of flow, on a length of 1000 mm. As for the direction *y*, 60 nodes are distributed on 20 mm of length, but with an initial height of the control volume of 0.01 mm between the first two nodes, while the distance between each subsequent two nodes increases with a geometric progression of 1.1 to the middle of the channel i.e., the so-called bigeometric progression. Like the *y* direction and in the *z* direction on the length of 50 mm, 100 nodes are arranged by bigeometric progression with a step of 1.15, where the height of the control volume between the first two nodes is 0.05 mm. The mesh thus formed gives 900,000 cells or control volumes and is given in Figure 3.

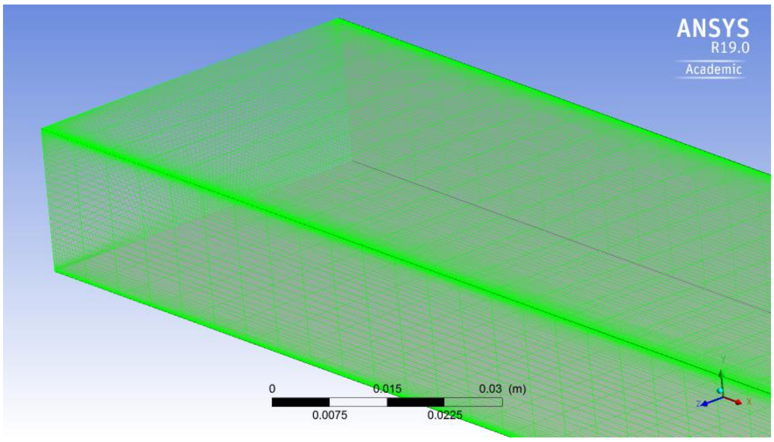

Fig. 3 Mesh model in ICEM-CFD

For the validation of the developed model of EMHD flow of micropolar fluid in ANSYS, the analytical model of EMHD flow of micropolar fluid between parallel plates, defined by equations (1) to (5) is used.

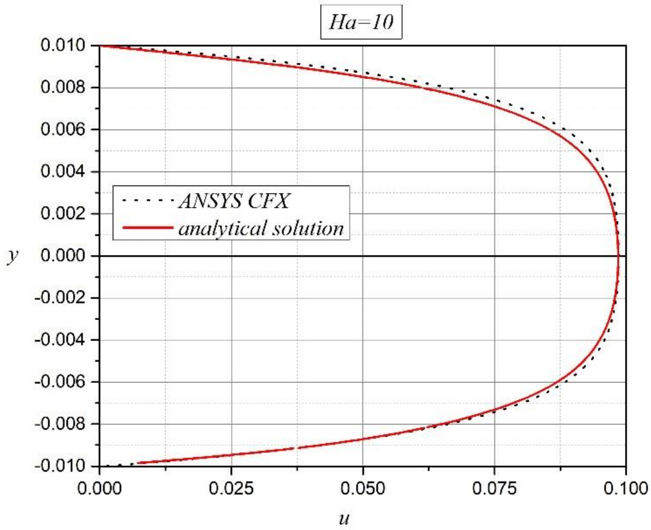

Fig. 4 Comparison of analytical and numerical solutions for laminar flow

It is obvious from Figure 4 that the velocity solution obtained based on CFD simulations converges quite well to the velocity solution obtained based on the analytical solution of the mathematical model of the EMHD micropolar fluid flow. The results obtained in this way confirm the validity of the model in ANSYS CFX, as well as the good density of the selected mesh for fast convergence. When performing simulations, the allowed error i.e., the RMS (root mean square deviation) size is defined to be less than 10^{-5}, and the number of iterations is determined to meet the given conditions.

3. RESULTS

The analysis of the obtained simulation results was performed through the influence of Hartmann's number *Ha*, i.e., the influence of the external magnetic field, then the influence of the external electric field or the load factor *Q* and finally the influence of the additional viscosity of the micropolar fluid, i.e., the influence of the coupling factor *K*. The analysis of the given characteristic parameters was performed on the profile of the velocity and microrotation of the micropolar fluid and is given in the following graphs.

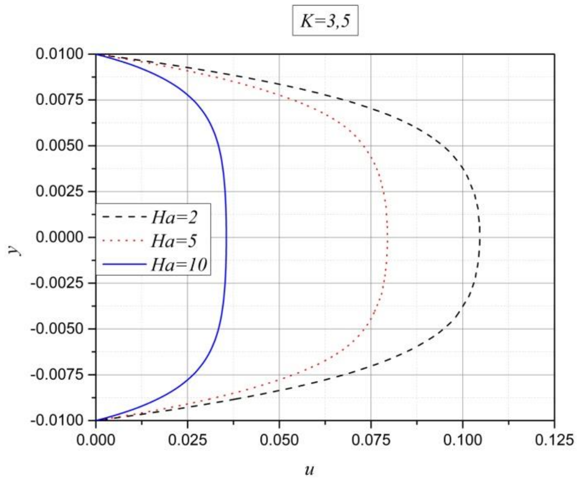

Fig. 5 The influence of the Hartmann number *Ha* on the velocity profile

The first two figures, Figure 5 and 6, represent the influence of the Hartmann number on the velocity and microrotation field.

From the given figures, it is clearly observed that the tendency of the change of velocity and microrotation with the increase of the Hartmann number corresponds to the results obtained during the analytical solution and analysis of the micropolar fluid flow between two plates investigated in the previous work [10]. The tendency of the change of velocity and microrotation is such that the increase in the Hartmann number leads to the reduction and alignment of the field of velocity and microrotation over the entire height of the channel. In addition, it can be concluded that the increase in the intensity of the magnetic field leads to a decrease in the characteristics of micropolar fluids.

The next two figures, 7 and 8, represent the influence of the external electric field, through the load factor *Q*, on velocity and microrotation. The analysis of the impact of load factor will be performed for three cases for values of load factor of -1, 0 and 1. It is well known that for value *Q*=-1 the system works as a flow meter, while for value *Q*>1 the system behaves as a pump. At the value of the load factor *Q*=0, there is no influence of the electric field, and it is important to note that during the analysis of the influence of the external electric field, the intensity of the external magnetic field was constant, i.e., the value of Hartmann's number was *Ha*=10.

From figure 7, at the value Q=1 there is a change in the direction of fluid flow, while at the value Q>-1 there is an increase in the intensity of velocity in relation to laminar flow. In figure 8, the influence of the load factor Q on the microrotation is the same as on the velocity profile. The results obtained in this way absolutely correspond to the analysis of the micropolar fluid flow between the plates, which was done in one of the previous manuscripts [10].

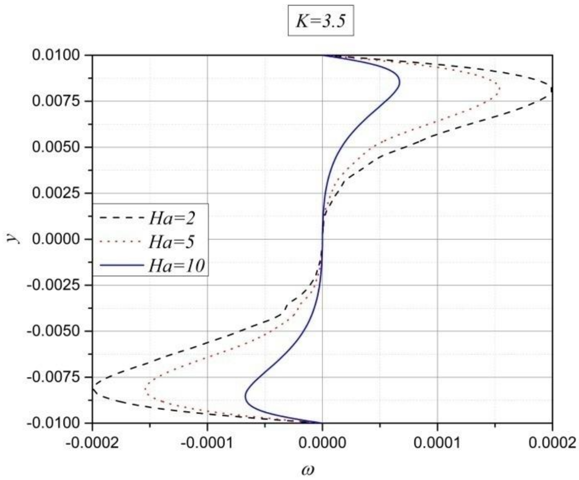

Fig. 6 The influence of the Hartmann number *Ha* on microrotation

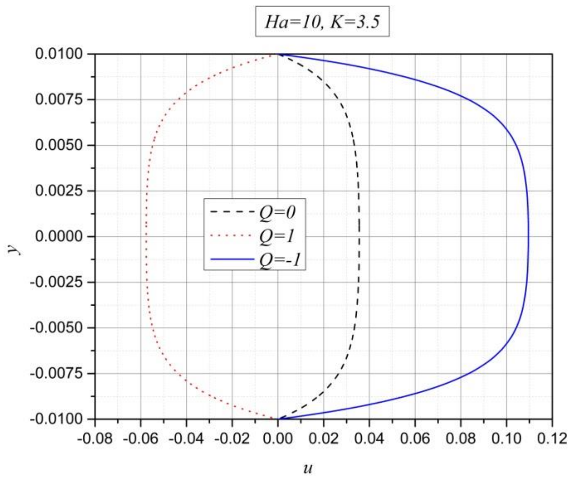

Fig. 7 Effect of load factor Q on velocity

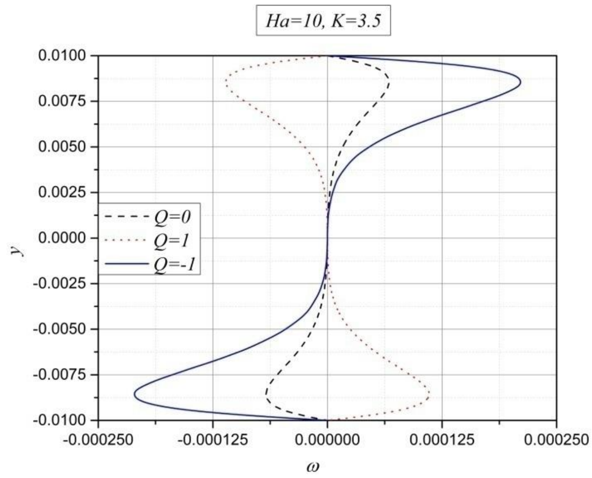

Fig. 8 Effect of load factor *Q* on microrotation

Finally, on the last two figures 9 and 10, it will be presented the influence of additional viscosity λ, i.e., of the coupling factor *K* to the velocity field and microrotation of the micropolar fluid.

Figure 9 shows the influence of the coupling parameter *K* on the velocity field. It can be seen from the graph that an increase in the coupling parameter *K* leads to a decrease in the fluid velocity. On the other hand, figure 10 shows that an increase in the coupling parameter *K* leads to an increase in the intensity of microrotation, which is expected because the increase in the coupling factor is a consequence of the increase in additional viscosity λ, which is one of the characteristics of a micropolar fluid, such as the vector of microrotation.

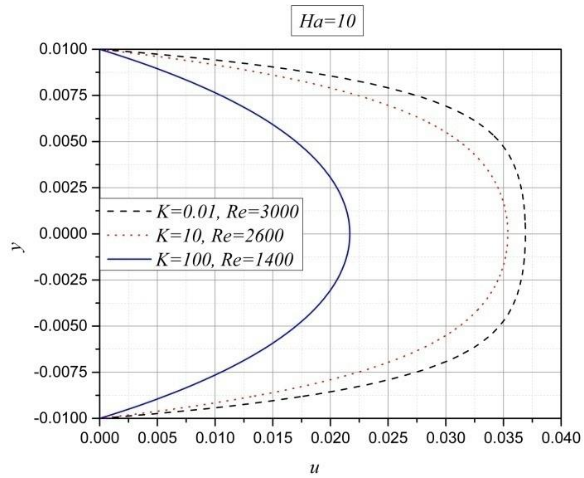

Fig. 9 Influence of the coupling factor *K* on the velocity profile

Like the previous four figures, the last two also give the same tendencies in the change of velocity and microrotation with the change of the coupling factor K as in the analytically considered model of micropolar fluid flow between the plates [10].

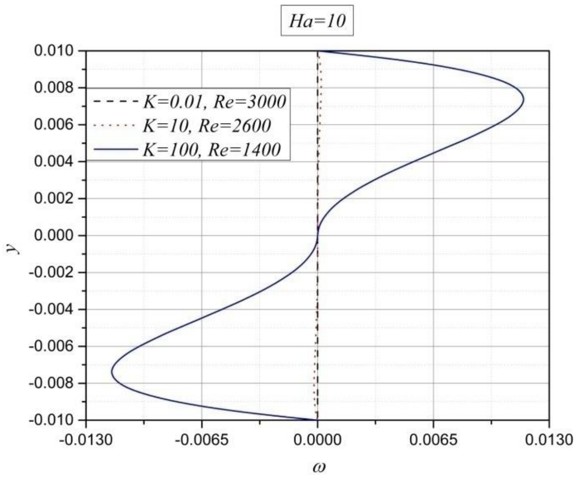

Fig. 10 Influence of the coupling factor *K* on the microrotation

The analysis and comparison of the results carried out in this way confirms that the formed numerical model in the ANSYS software package for micropolar fluid flow is good, and that it can be used as such in the further analysis of micropolar fluids.

The main advantage of modern software is the speed of data processing and problem solving, as well as the accuracy of the obtained results. However, one should not underestimate another advantage offered by modern software for the analysis of flow and heat transfer and that is high-quality processing and visualization of the results. The ability to display the obtained results in color provides great advantages, such as a large number of data displayed in different colors depending on the intensity of the displayed quantity, which gives a very good insight into the flow and thermal processes taking place in the observed system. The next couple of pictures will serve to present the results obtained during the analysis of the MHD flow of a micropolar fluid in a horizontal channel. In the next two figures 11 and 12, the pressure changes along the direction of flow, as well as the velocity field in the initial part of the channel, are presented.

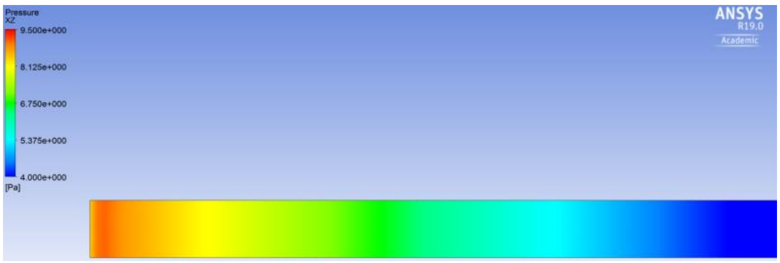

Fig. 11 Change in pressure along the flow direction

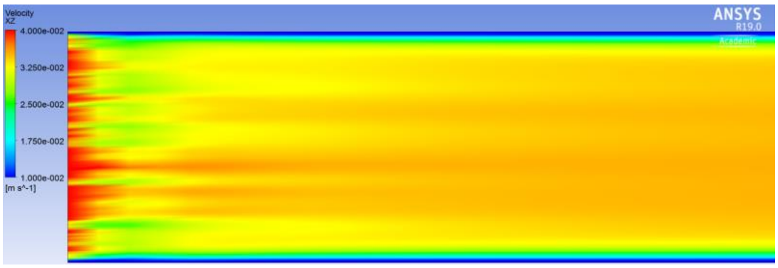

Fig. 12 Velocity field at the beginning of the channel

In addition to presenting the results using a color palette, it is also possible to do this with the help of a vector display. The next figure 13 shows the velocity vector in the form of 3D arrows.

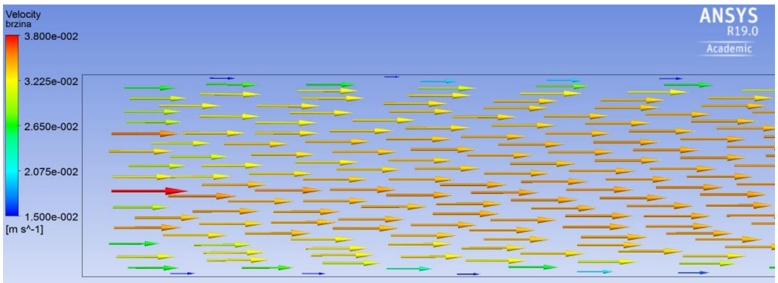

Fig. 13 Velocity field – vector display

The analysis of the influence of the magnetic field on the flow velocity in the channel can best be seen in the following figure 14, where the profile of the velocity at the very entrance to the channel and the profile of the velocity after the effect of the magnetic field are presented in parallel.

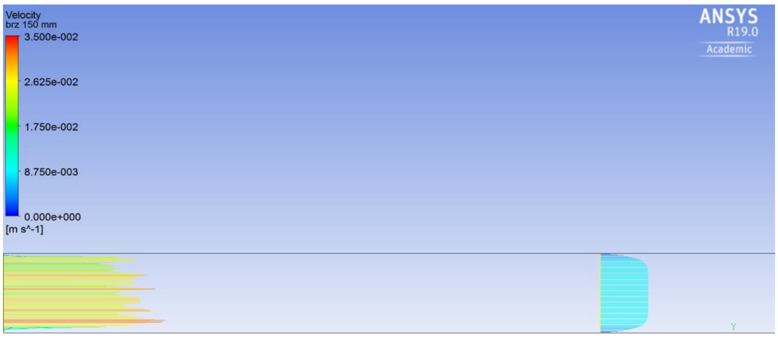

Fig. 14 The influence of the magnetic field on the velocity profile

4 CONCLUSION

Since the micropolar fluid model is not offered in the palette of available fluids, the aim of the work was to create a model for the investigation of micropolar fluid flow in one of the modern software packages. As the previous analyzed model in ANSYS CFX, EMHD flow of micropolar fluid in a horizontal channel, gave results in which the influence of characteristic parameters has the same tendency as in the analytical model of planar EMHD flow of micropolar fluid between two plates, we can consider that the developed model simulates the behavior of these fluids quite well, and that such a model can be applied to describe some other types (models) of micropolar fluid flow. Certainly, further research and improvements on the model must be carried out, so that in the future the palette of available types of fluids includes a micropolar fluid, which has widespread application.

Acknowledgement: *This research was financially supported by the Ministry of Education, Science and Technological Development of the Republic of Serbia (Contract No. 451-03-47/2023-01/200109).*

REFERENCES

1. J. C. Tannehill, D. A. Andreson, R. H. Pletcher, 1997, *Computational fluid mechanics and heat transfer*, Taylor & Francis, Second edition, Washintong D.C., 803 p.
2. Tu J., 2019, *Computational fluid dynamics a practical approach*, 3rd edn. Elsevier Ltd, Amsterdam, 498 p.
3. Arkadiusz S., Weronika B., Iwona W. K., Salih M. S., Andrzej P., 2023, *Application of computational fluid dynamics simulations in food industry*, European Food Research and Technology, 249, pp. 1411–1430.
4. Jagadale P., Chawdhary A.B., 2021, *Computational fluid dynamics, an overview,* International Research Journal of Engineering and Technology, 8, pp. 1817–1821.
5. Ernst H. H., Egon K., 2009, *100 Volumes of 'Notes on Numerical Fluid Mechanics', 40 Years of Numerical Fluid Mechanics and Aerodynamics in Retrospect,* Springer, Berlin, 498 p.
6. Grzegorz L., 1999, *Micropolar Fluids – Theory and Applications,* Springer, New York, 253 p.
7. A. C. Eringen, 1966, *Theory of micropolar fluids*, Journal of Mathematics and Mechanics, 16, pp. 1 – 18.
8. M. Kocić, Ž. Stamenković, J. Petrović, J. Bogdanović Jovanović, 2023, *Control of MHD Flow and Heat Transfer of a Micropolar Fluid through Porous Media in a Horizontal Channel*, Fluids, 8(3), 93.
9. M. Kocić, Ž. Stamenković, J. Petrović, J. Bogdanović Jovanović, 2023, *MHD micropolar fluid flow in porous media*, Advances in Mechanical Engineering, 15(6).
10. M. Kocić, Ž. Stamenković, J. Petrović, M. Nikodijević, 2018, *EMHD Micropolar Fluid Flow and Heat Transfer in a Channel*, The 4th international conference mechanical engineering in XXI century, April 19-20, 2018, Faculty of Mechanical engineering, University of Nis.
11. ANSYS Fluent Workbench Tutorial Guide, 2021, ANSYS, Inc. Release 2021 R2.
12. ANSYS Modeling and Meshing Guide, 2005, ANSYS, Inc. Release 10.0.
13. F. Alauzet, A. Loseille., 2016, *A Decade of Progress on Anisotropic Mesh Adaptation for Computational Fluid Dynamics*, Computer-Aided Design, 72, pp.13-39.