doi:10.22306/atec.v6i3.86 *Received: 08 June 2020; Revised: 06 Aug. 2020; Accepted: 04 Sep. 2020*

INVESTIGATING THE RESTRICTIVE EFFECT OF NEIGHBORING PERFORATION CHANNELS WITH COMPUTATIONAL FLUID DYNAMICS SIMULATION

Ádám V. Pásztor
Petroleum Engineering Department, University of Miskolc, Egyetemváros Miskolc, 3515 Miskolc, Hungary, padamv91@gmail.com (corresponding author)
Zoltán Turzó
Petroleum Engineering Department, University of Miskolc, Egyetemváros Miskolc, 3515 Miskolc, Hungary, turzoz@uni-miskolc.hu

Keywords: CFD simulation, effect of perforation design, flow in porous media, pressure distribution around perforations
Abstract: The inflow performance relationship of a well establishes the link between the applied pressure drawdown and the inflow rate at the bottom of the well. In a cased and perforated well the perforation parameters have a large effect on the inflow performance relationship. When investigating the effect of the different parameters, the most challenging task is to describe the connection between the phase angle of the perforation design and its performance. One of the authors of this paper published a method, which incorporates the perforation channels' restricting effect on the drainage area of each other, which is the function of the phase angle. The aim of this study to validate the restricting effect with the use of computational fluid dynamics (CFD) simulations.

1 Introduction

1.1 Perforation parameters

The perforation channels initiated by a device called perforator gun, which is technically a steel pipe containing many perforator heads filled with some kind of explosive. The perforation process is the conveying of this perforator gun to the desired depth and the firing the explosive (Figure 1).

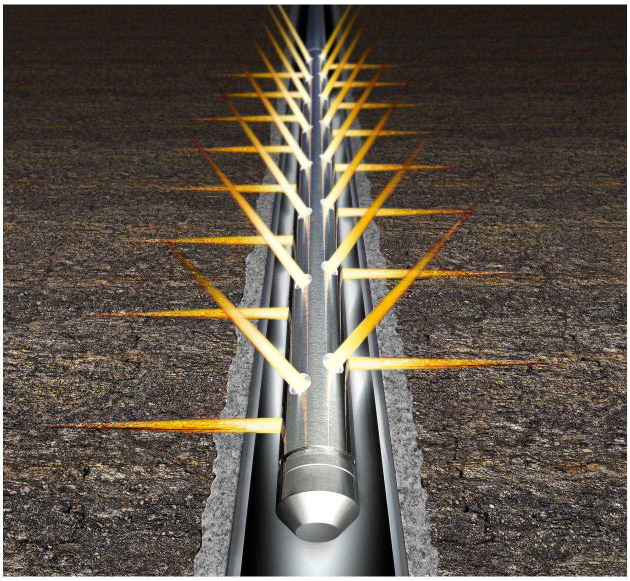

Figure 1 Perforation process
[Source: https://www.halliburton.com/en-US/ps/wireline-perforating/wireline-and-perforating/perforating-services/CHE-System.html, viewed 03 December, 2014]

The perforation design, which is shown on the figure below (Figure 2), is the result of the length of the gun, the type and spatial distribution of the perforator heads and the explosive's type and quantity.

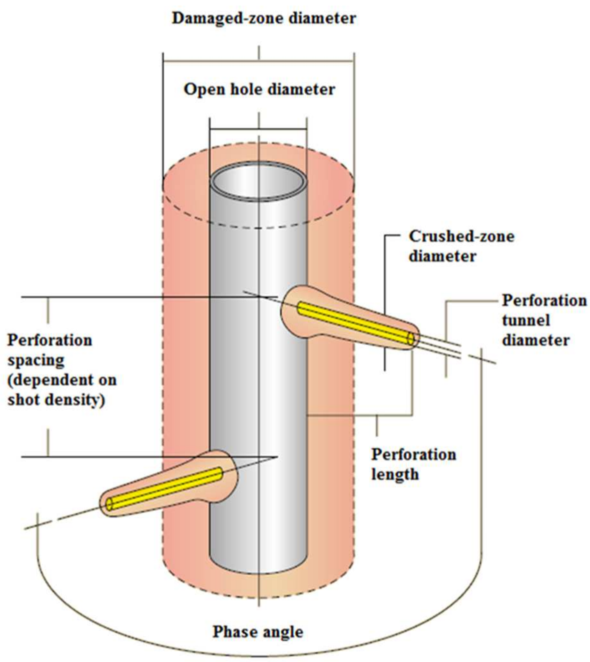

Figure 2 Perforation parameters [1]

The different parameters can be altered by modifying the perforator gun. The commercially available phase angles are presented on the Figure 3.

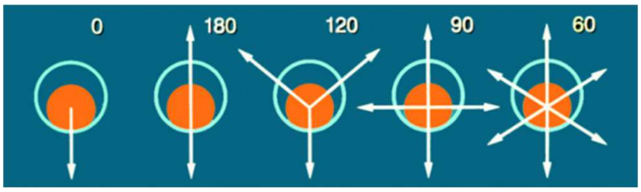

Figure 3 Available phase angles
[Source: https://petrowiki.org/Perforating_design, viewed03 December 2019]

1.2 IPR models for perforated wells

The Inflow Performance Relationship (IPR) of a well describes the change in the inflow in function of the bottom hole pressure, thus it is the perfect mean to investigate the efficiency of a production system.

The relationship between the perforation parameters and the performance of the well is investigated since the 1980s. Yldiz carried out a large-scale investigation on the different available calculation method by checking their accuracy with measured rate independent skin factors [2]. He pointed out, that the two most widely used methods are the method of Karakas and Tariq [3] and the method of McLeod [4].

Karakas and Tariq presented a semi-analytical solution for the skin calculation in perforated completions. They have quantified the wellbore and vertical-flow effects by finite-element simulations. They have showed that the skin effect which is caused by perforations is a combination of four different effects as follows: horizontal skin, vertical skin, wellbore skin, and crushed zone-effects.

McLeod assumed that perforations are small wells in the reservoir and used the Jones [5] method to calculate the pressure drop across them. Unfortunately, this method does not take the effect of phase angle into consideration.

Pásztor and Schultz [1] pointed out in their paper that both of the previously presented methods have some contradictions in their results, thus their application for calculating the IPR of perforated wells is not advised. Based on their further investigation they introduced a new model, which doesn't have the contradictions of its predecessors. Their base concept was to determine the drainage volume around the perforation channels and calculate the productivity of each individual channel accordingly.

2 Restrictive effect of perforation channels

During the filtration from the reservoir to the wellbore there is a well identifiable change in the flow direction from perpendicular to the axis of the well to be perpendicular to the axis of the nearest perforation channel (Figure 4).

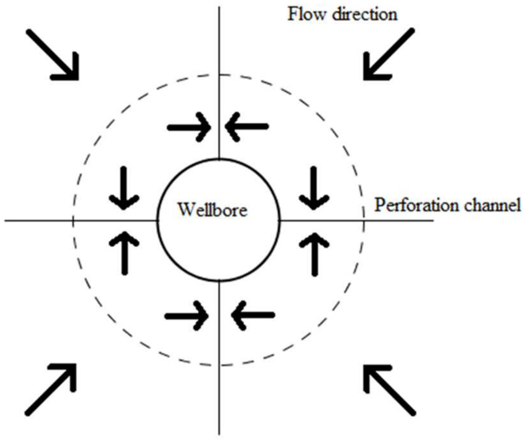

Figure 4 Change in flow direction

The pressure drop during the filtration is proportional to the drainage radius i.e. the furthest point from which fluid flows to the draining point. To the point of direction change this drainage radius is actually the wells drainage radius and the flow to this point is actually a flow to a pseudo well with higher wellbore radius.

After the direction change a flowing particle will flow towards the perforation channel which is the closest to it. The points which are closer to a given perforation than any other, constitutes the drainage volume of the given channel. The closer are the channels to each other the smaller is this volume. This behavior is called the restrictive effect of the perforation channels.

Geometry of the perforations' drainage volume

A perforation channel has six neighboring perforations around it, thus at any given distance from its base there are six points in the perpendicular plane which are equally far from an other perforation. These six points will define an ellipse, which is the boundary of the drainage area at the plane perpendicular to the axis (Figure 5).

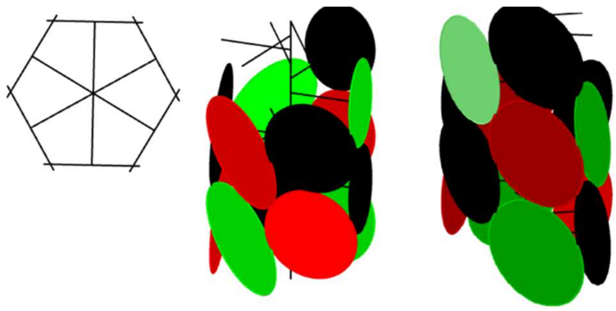

Figure 5 Restrictive effect of perforation channels

The perforation channels are actually evasive straight lines, thus the shape of the drainage space is relatively complex as shown in the figure below (Figure 6).

Volume: 6 2020 Issue: 3 Pages: 79-84 ISSN 2453-675X

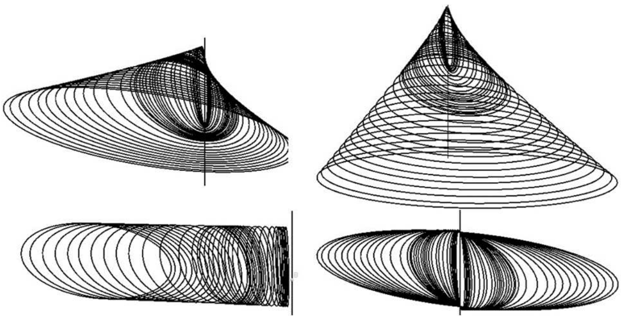

Figure 6 Drainage space of perforation channels

3 CFD model

3.1 Introduction

Computational fluid dynamics techniques provide a way for determining the characteristics of fluid flow around the perforated interwall of a fluid producing well. CFD calculations numerically solve the governing equations for a flowing fluid. To facilitate the simultaneous solution of the governing equations, the procedure involves dividing the flow space into sufficiently small finite volumes or cells [6,7]. The accuracy of flow modeling greatly depends on the proper setup of these cells. This article describes this spatial distribution.

After one properly sets up the cell structure, CFD calculations can determine the pressure distribution, the flow path and flowing velocities etc. in the evaluated domains.

Based on the CFD results, we are validating the assumptions about the change in flow directions and the presence of restrictive effect.

3.2 Basic flow equations

Generally, during CFD simulations, to model the fluid flow the following equations are used.

The Navier-Stokes equation representing Newton's second axiom characterizes flow inside regular fluid domains (flow in . (Equation 1).

$$\frac{\partial \vec{v}}{\partial t} + Div(\vec{v} \circ \vec{v}) = \vec{g} - \frac{1}{\rho} grad p + \vec{v}\Delta\vec{v} + \frac{\mu + \zeta}{\rho} grad\ div\vec{v} \quad (1)$$

where: $\vec{v}$ = velocity vector,
$\vec{g}$ = acceleration gravity vector,
$\boldsymbol{\rho}$ = fluid density,
$\boldsymbol{p}$ = pressure,
$\boldsymbol{\mu}$ = dynamic viscosity,
$\boldsymbol{\zeta}$ = specific viscosity,
$\boldsymbol{t}$ = time.

The continuity equation, Equation 2, describes the conservation of mass, while Equation 3 expresses the conservation of the energy of the flowing fluid.

$$\frac{\partial \rho}{\partial t} + div(\rho\vec{v}) = 0 \quad (2)$$

$$\frac{\partial}{\partial t}\left(\frac{v^2}{2} + h\right)\rho + div\left[\left(\frac{v^2}{2} + h\right)\rho\vec{v} - \lambda\nabla T\right] = \frac{\partial p}{\partial t} \quad (3)$$

where: $\boldsymbol{h}$ = thermodynamic enthalpy,
$\boldsymbol{\lambda}$ = heat conductivity,
$\boldsymbol{T}$ = temperature.

Because these three basic differential equations governing the flow conditions have five unknowns, an unambiguous solution requires two more equations. These are the equation of state for the flowing fluid and the equation describing the change of the enthalpy with the state parameters (Equations 4 and 5).

$$\rho = \rho(T, p) \quad (4)$$

$$h = h(T, p) \quad (5)$$

The five equations constitute a system of equations with five unknowns, making the simultaneous solution of them theoretically possible.

The flow around the well bore is situated in a porous of reservoir rocks (in porous medium) until it reaches the perforations and the well bore. The porous model is the combination of Darcy's law (Equation 6.) commonly used for flows in porous regions and a generalization of the Navier-Stokes equations. This combination can be used to simulate flows if the geometry is too complex to resolve with a grid. The model retains both advection and diffusion terms.

$$\nabla p = -\frac{\mu}{k}\vec{v} \quad (6)$$

where: k = absolute permeability.

During deriving the continuum equations, ‘infinitesimal’ control volumes and surfaces, large relative to the interstitial spacing of the porous medium, but small relative to the scales wanted to be resolved was assumed. The given control cells and control surfaces are assumed to contain both solid and fluid regions.

3.3 Numerical solution

Because an analytical solution of the basic flow equations (Equations 1-6) is possible only for very simple cases, complex cases usually require numerical solutions, so-called finite element models.

The computational fluid dynamics program package used in this article performs a numerical solution of the governing equations using finite volumes. The program constructs these by dividing the flow space into a finite number of cells with finite volumes connected to each other.

It makes calculations at the geometrical centers of the cells (the node points) and first calculates the fluid and flow parameters. Then it solves the algebraic equations resulting from the integration of the basic differential equations at cell boundaries to obtain the five unknowns at the node points.

3.4 Geometrical flow space model

Results of CFD calculations depend on the proper setup for the geometrical model of the flow space because improper models can cause convergence difficulties and erroneous results. Figure 7a and b. shows the geometrical model of the CFD simulation. In our case, the geometrical model means the representation of reservoir rock around the well and the perforation channels and the well bore. The reservoir rock is a porous domain. The porous domain has cylindrical shape, and in its center the wellbore is located. Both the wellbore and the perforations have cylindrical shapes. The reservoir rock is furtherly divided into two zones. The outer one, the unaltered rock zone. It represents the original reservoir rock. The inner one, the damaged zone, representing the damaged zone due to drilling process. The damaged zone has smaller permeability and porosity than the unaltered zone. There are eight perforations in the model, the phase angel is 90 °. The length of perforations is greater than the damaged zone thickness, it reaches the middle of the unaltered zone, the vertical distance between the perforations at the same angle positions are different for the different angles, to show the effect of it on the inflow behavior. As the figures shows, the whole flow domain is furtherly divided into smaller blocks to help the proper meshing process.

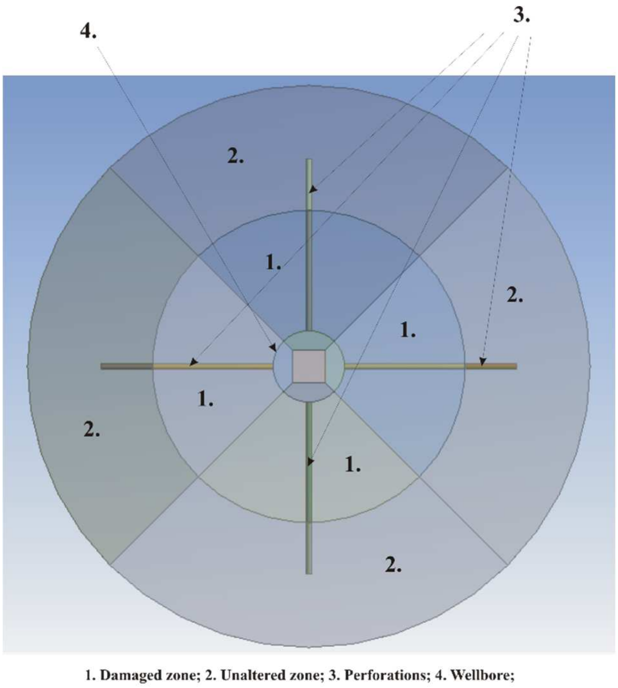

Figure 7a Top view of model geometry

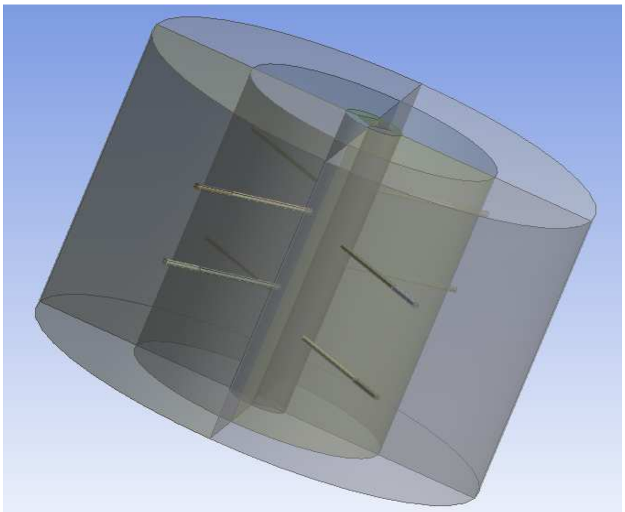

Figure 7b: Isometric view of model geometry

The calculations require the filling of this space with interconnected hexahedrons representing the cells. Ensuring higher accuracies and faster solutions requires selection of the hexahedrons that are nearly cubes.
Figure 8. shows the mesh structure of the fluid domain. One block is hidden on the figure, to show the mesh around perforations, too. The mesh is generated by hex dominant method and includes 491680 elements.

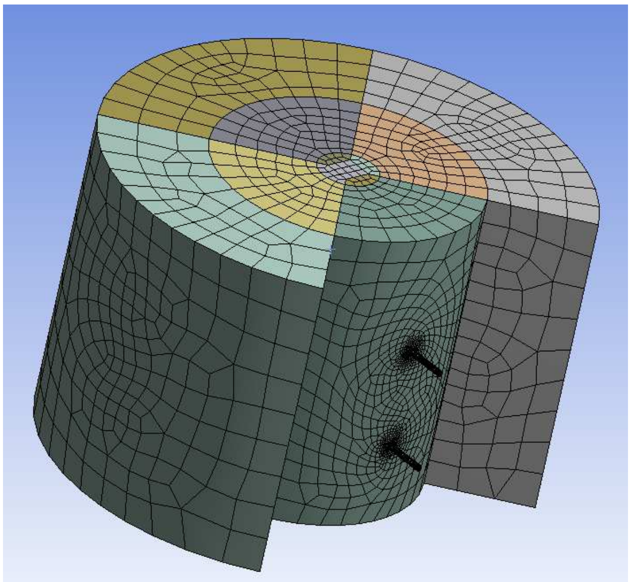

Figure 8 Mesh structure of the model

3.5 Simulation results

The aim of the CFD simulation was to prove the flow direction change near the perforations and to show the restrictive effect of the neighboring perforations. No quantitative analysis made on flow rates.

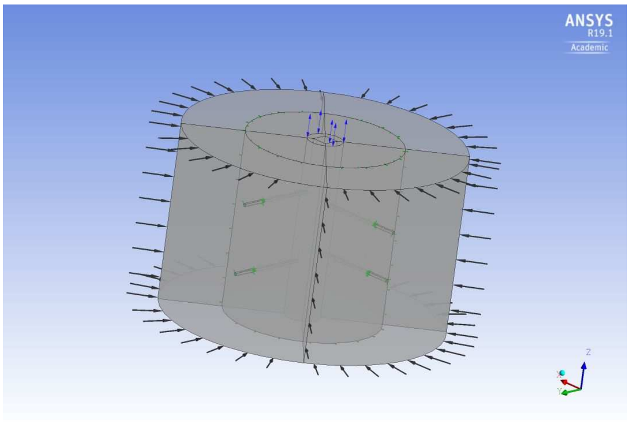

Figure 9 Inlet and outlet ports of the model

Steady state, isotherm flow model with constant pressure boundaries used at inlet and outlet of the flow domain. The Figure 9. shows the inlet and outlet ports of the model. The inlet indicated by black, while outlet by blue arrows. Water flow was modelled while the porous domain 100% saturated by water. Isotropic loss model selected neglecting inertial losses.

The most important results of the simulation for our purpose are the spatial pressure distribution and flowing directions in the model. Figure 10. shows the pressure distribution around perforation channels rendered on some selected planes.

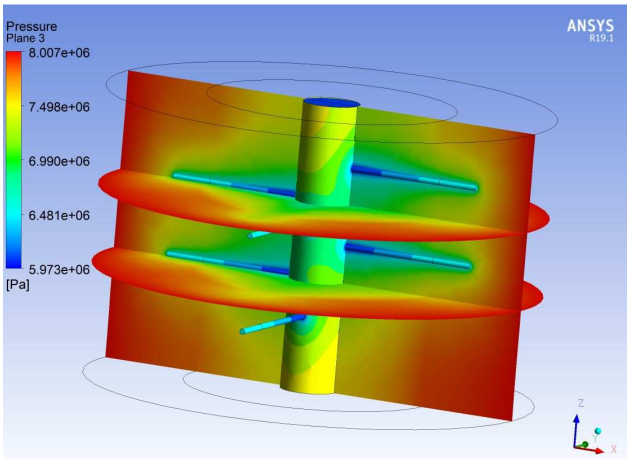

Figure 10 Pressure distribution around perforation channels

Evaluating the results on the figure one can state that the pressure is decreasing towards the perforations. The pressure gradient is higher in the damaged zone than in the unaltered zone. The pressure loss is smaller in the perforations. The pressure profiles around perforations are altered by neighboring perforations. This is the results of restriction effect. The effect is higher for the perforations are closer to each other.

Plotting iso-pressure surfaces around perforations shows the restricting effect much better and illustrates the shape of the flow space around perforations. Figure 11 demonstrate one of the iso-surface of pressure.

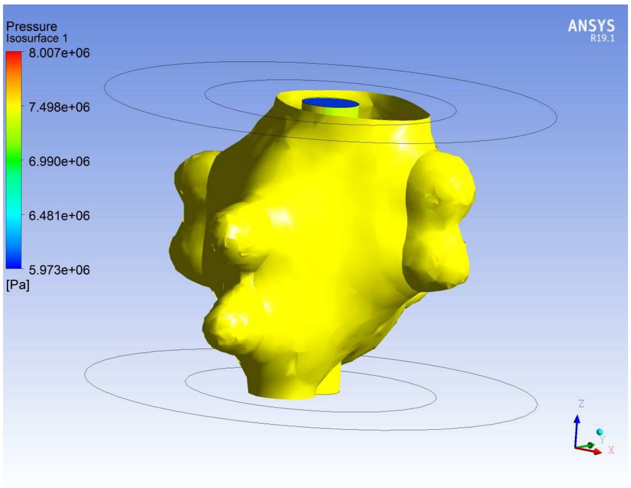

Figure 11 Iso-surface of pressure

Evaluating the shape of the iso-surface, is clear that the restrictive effect is prevail and it is affected by the distance between the perforations. The form of the iso-surface on end of perforations toward the inflow boundary looks like an ellipsoid. It is in sync with our previous statement about elliptical shape of drainage area of perforations.

Another important aspect of the new inflow model for perforated well is the assumption of flow direction change around the perforations. To show the flow directions in the flow space the most convenient way is plotting the streamlines (trajectory) of the flow.

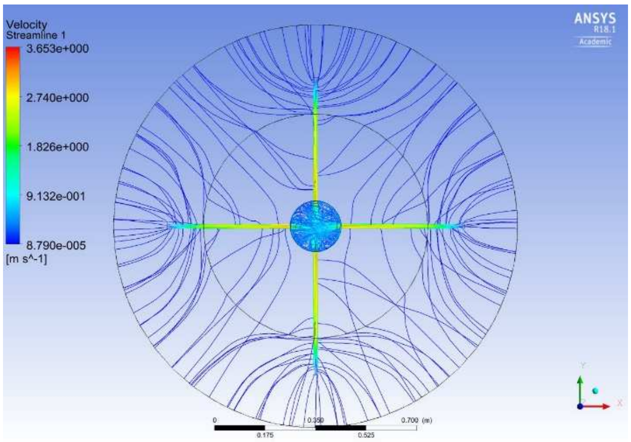

Figure 12 Change in flow direction CFD

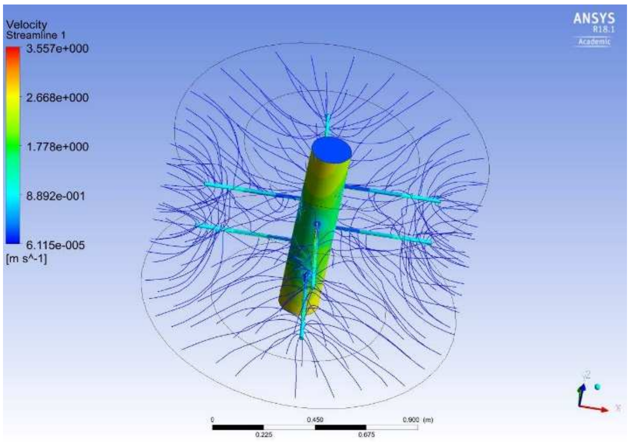

Figure 13 Restrictive effect of perforation channels CFD

Figure 12 and 13. show the propagation of streamlines around the well from two different viewpoints. The streamlines started from the inflow boundary. On the figures, it clearly shown that the flow directions changing around the perforations. The directional change can be identified in both the unaltered and damaged zones. The streamline density is greater in the undamaged zone. It indicates higher flow rates there.

4 Conclusion

The method proposed by Pásztor & Schultz introduced the restrictive effect of neighboring perforation channels as a new concept for calculating the pressure drop of perforates wells. The application of computational fluid dynamics simulation was proven to be the perfect tool to check the validity of the concept.

The results of the CFD simulation provided a series of visual representation of the flow behavior around the perforation channels. The pressure distribution and the pressure iso-surface are good representations of the restrictive effect of adjacent perforation channels. Furthermore, the shape and density of the streamlines and the shape of the pressure iso-surface describe the shape of the drainage area around the perforation channels as well.

- The pressure profiles around perforations are altered by neighboring perforations, which is indicated by the change in the pressure distribution between the perforations.
- The effect is higher for the perforations are closer to each other, which can be seen from the density difference in the streamlines between the vertical and horizontal axis.
- As the flow is perpendicular to the pressure iso-surface, the results are in sync with our previous statement about elliptical shape of drainage area of perforations.

The presented results prove the existence of the restrictive effect and also validates the concept of Pásztor and Schultz about the shape of the drainage areas.

The density of the streamlines is different in the altered and unaltered zones. This behavior indicates that vast portion of the fluid enters the perforation channels in the unaltered zone. This phenomena is crucial regarding the productivity intensification of the wells and its theoretical background was discussed by Pasztor in 2016 [8].

The further investigation of the effect of the altered zone on the flow around the perforated production well with CFD simulations will be the continuation of this research.

References

[1] PASZTOR, A., SCHULTZ, V.: Analytical determination of the perforation design’s effect on the productivity, *MOL GROUP Professional Journal*, Vol. 2016, No. 1, pp. 108-117, 2016.
[2] YLDIZ, T.: Assessment of Total Skin Factor in Perforated Wells, *SPE Reservoir Evaluation & Engineering*, Vol. 9, No. 1, pp. 61-76, 2016.
[3] KARAKAS, M., TARIQ, S. M.: Semi-analytical Productivity Models for Perforated Completions, *SPE Production Engineering*, Vol. 6, No. 1, pp. 73-82, 1988.
[4] MCLEOD, O. H. J.: The Effect of Perforating Conditions on Well Performance, *Journal of Petroleum Technology*, Vol. 35, No. 1, pp. 31-39, 1983.
[5] JONES, L. G., BLOUNT, E., GLAZE, O. H.: *Use of Short-Term Multiple-Rate Flow Test to Predict Performance of Wells Having Turbulence*, SPE Annual Technical Conference and Exhibition, New Orleans, 1967.
[6] FERZIGER, J., PERIĆ, M.: *Computational Methods for Fluid Dynamics*, Berlin, Spriger, 2002.
[7] FLETCHER, C.: *Computational Techniques for Fluid Dynamics 1*, Berlin, Springer, 1998.
[8] PÁSZTOR, Á.: *The Invaded Zone’s Effect on The Inflow of Perforated Wells*, SPE Hungarian Section, 17. November 2016, [Online] Available: http://hungary.spe.org/viewdocument/spe-hun-conference-2016. [10. Apr. 2020], 2016.

Review process

Single-blind peer review process.