METHODS OF CFD MODELLING OF TWIN-SCREW PUMPS FOR NON-NEWTONIAN MATERIALS

MARIAN BOJKO, LUKAS HERTL, SYLVA DRABKOVA

VSB Technical University of Ostrava, Faculty of Mechanical Engineering, Department of Hydromechanics and Hydraulic Equipment, Ostrava, Czech Republic

DOI: 10.17973/MMSJ.2021_12_2021103

marian.bojko@vsb.cz

The twin-screw pump is designed for pumping highly viscous materials in the food industry. Rheological characteristics of materials are important in the specification of design parameters of screw pumps. Analysis of flow in the twin-screw pumps with definition of non-newtonian materials can be made by numerical modelling. CFD generally oriented software ANSYS Fluent and ANSYS Polyflow has been used for modelling. In this study those software's (ANSYS Fluent and ANSYS Polyflow) were defined for solution of flow in the twin-screw pumps. Results were compared for the same boundary conditions on the inlet and outlet of the 3D model. For definition of the viscosity were used the Nonnewtonian power law. Parameters as consistency coefficient and flow exponent for Nonnewtonian power law were analysed by software ANSYS Fluent and ANSYS Polyflow. Postprocessing form ANSYS Fluent and ANSYS Polyflow were made by contours of field and by graphs.

KEYWORDS

TWIN-SCREW PUMP; CFD ANALYSIS; ANSYS; NON-NEWTONIAN MATERIAL; POLYFLOW; OVERSET

1 INTRODUCTION

The screw pumps are primarily used in transport of highly viscous materials in the food industry. Most of the published articles on the topic of screw pumps are related to the problematic of polymer extrusion or the food industry, specifically the transport of biomass and food mixtures. Screw pumps are generally divided into one, two and three spindle screw pumps. In this case, a twin-screw pump is considered (Figure 1). Twin screw pumps can be designed to rotate either in the same direction (co-rotating screws) or in opposite directions (counter-rotating screws). Therefore, the research of high-viscosity mass transport is demanding in terms of CFD numerical calculations because it is necessary to define the rotation of the relevant objects according to the available numerical methods. ANSYS Polyflow or Fluent software with remeshing technology is usually used for CFD simulations in articles on this topic. The issues of CFD analysis of a twin-screw pump with co-rotating and counter-rotating screws is handled by ANSYS Polyflow software in articles [Wilczynski, 2016], [Lewandowski, 2017]. The effect of screw pitch is particularly evaluated in the publications. Non-Newtonian fluids are considered for the transport of highly viscous materials, where the viscosity is defined mainly by the power function in CFD analysis. The influence of the viscosity definition by the power function in the calculation of fluid transport by screw pump in the ANSYS Fluent is the subject of a publication [Tagliavini, 2016]. Another area of research associated with the transport of fluids by screw pump is the evaluation of radial forces acting on the spindles. The methodology of force evaluation based on CFD analysis is the content of the publication [Ryazantsev, 2010].

The aim of this work was to apply a new method of solving screw pumps by using overset technology contained in the Fluent software and to compare the results with one of the already proven simulation method.

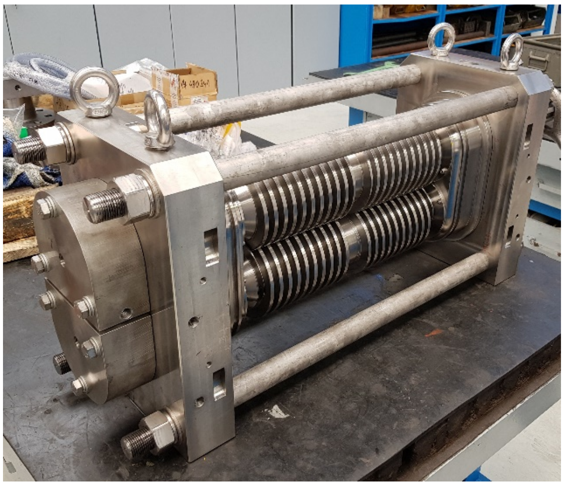

Figure 1. Manufactured twin-screw pump

2 MODEL OF SCREW PUMP

For the purposes of CFD simulations, a simplified geometric model of a twin-screw pump was created based on a real device manufactured by Hydrosystem project a.s. company under project number FV40105 (see Figure 1).

Two spindles with a shaft diameter of 59.9 mm with counter rotation were modelled. Scheme of model you can see in the Figure 2. Each spindle has a thread with a pitch of 20 mm and with the number of 3.5 threads. The thread profile is trapezoidal. The stator chamber for the spindles has a diameter of 82 mm. The gap between the thread and the stator has been increased to 2.5 mm. In the case of a real twin-screw pump device, the size of gap can is usually in the range of 0.5 mm – 1 mm. The reason for increasing the gap in the model of the pump is to simplify the subsequent CFD analysis with regard to the creation of a mesh. The final CFD model will have a smaller number of cells so the calculations will not be so time consuming. This is desirable in the first phase when it is necessary to test the suitability of the defined methodology for problem solution.

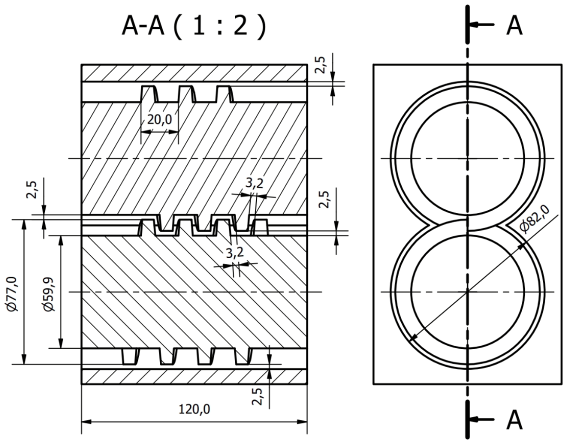

Figure 2. Dimensional scheme of created twin screw pump

In terms of thread profile, square profiles are also used in addition to the trapezoidal. Also pitch of 40 mm can be considered. The general parameters of the design of screw pumps are the type of thread profile, pitch, width and height of the thread.

3 CFD SIMULATION OF FLUID TRANSPORTATION BY SCREW PUMP

The mathematical model for isothermal fluid flow is generally defined by the basic equilibrium equations expressing the fundamental laws.

Continuity equations expressing the law of conservation of mass in differential vector form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{u}\right) = 0 \tag{1}$$

for general space-time dependent flow.

Furthermore, the Navier-Stokes equations expressing the law of conservation of momentum:

$$\frac{\partial \left(\rho \vec{u}\right)}{\partial t} + \nabla \cdot \left(\rho \vec{u} \vec{u}\right) = -\nabla p + \rho \vec{a} + \nabla \cdot (\tau) \tag{2}$$

for a general space-time dependent flow.

The computational domain for which the above system of equations will be solved can be constructed in many ways. In this case, the method of overlapping meshes was used, where the computational domain is meshed only once at the beginning and then the position of the moving mesh is updated for each time step. In this way, complex geometries can be meshed and conservative time and hardware intensive remeshing of the entire computational domain for the transient simulations can be avoided.

ANSYS Fluent software was primarily used for numerical calculations. Subsequently, ANSYS Polyflow software was used to compare the results from the ANSYS Fluent software. The obtained results were compared for the same boundary conditions. Overset technology was used in the case of ANSYS Fluent software. In the case of Polyflow, mesh superposition technique was used.

3.1 ANSYS Fluent mathematical model

The numerical solution problems for highly viscous materials in the spatial domain in ANSYS Fluent use the basic balance equations Equation 1 and 2 as mentioned in the documentation [ANSYS Fluent, 2020]. Overset technology, also sometimes referred to as 'Chimera', is used to create the domain. This is a method to easily create a domain for CFD calculations even for relatively complex geometries by first creating a clean mesh of fluid environment. This mesh is referred to as 'Background Mesh'. Subsequently, separate meshes are created for the surroundings of the individual parts/components/details that are surrounded by the fluid. These meshes are referred to as 'Component Meshes'. The resulting overall computational domain is created by overlaying the individual component meshes with the background mesh according to the overset interface settings. The data is then interpolated at the overlapping mesh locations while solving the system of equilibrium equations. This technology is particularly suited for solving complex geometry that undergoes certain motions.

3.2 ANSYS Polyflow mathematical model

ANSYS Polyflow uses the Mesh Superposition Technique (MST) when solving problems involving transient flows with moving objects. In the calculation of these problems with internal moving parts, modified Navier-Stokes equations are used in contrast to ANSYS Fluent [ANSYS Polyflow, 2020]:

$$H(\boldsymbol{v} - \overline{\boldsymbol{v}}) + (1 - H)(-\nabla p + \nabla \cdot \boldsymbol{T} + \rho \boldsymbol{g} - \rho \boldsymbol{a}) = 0 \tag{3}$$

A modified mass conservation equation is also used:

$$\nabla \cdot \boldsymbol{v} + \frac{\beta}{\eta} \Delta p = 0 \tag{4}$$

The Navier-Stokes equations are extended by the *H* parameter. This is a jump function that indicates by a value of 1 or 0 whether a given node is outside or inside the moving spindle geometry. Associated with this function is a residual factor that indicates whether the whole element will be counted as part of the flow domain or the moving part. The mass conservation equation is modified by the relative compression factor. How these factors affect the computational domain or the results can be found in [ANSYS Polyflow, 2020].

4 APLICATION OF CFD SIMULATION ON PREPARED 3D MODEL

The problem of transport of highly viscous materials specified in Chapter 1 was solved on a simplified 3D model (see Chapter 2) in the software ANSYS Polyflow 2020 R2 and ANSYS Fluent 2020 R2. A new geometrical part was created for the calculation, representing the fluid in the gap between the spindles and the stator. This geometry was meshed by a regular hexahedral mesh. This mesh has the same shape for both software solutions (see Figure 3).

The geometry of the spindles was modified for the calculations in ANSYS Polyflow. The material inside the shafts was mostly removed from the spindles, as the shafts are considered to be absolutely rigid for the calculation. After that, the spindles were meshed with tetrahedral elements (Figure 4). This step significantly reduced the total number of elements. This procedure has already been used in work [Xu 2018]. The prepared computational domain has 1,500,000 elements. Two another meshes of fluid with different number of cells were created to compare the effect of fluid mesh size on the results in Polyflow.

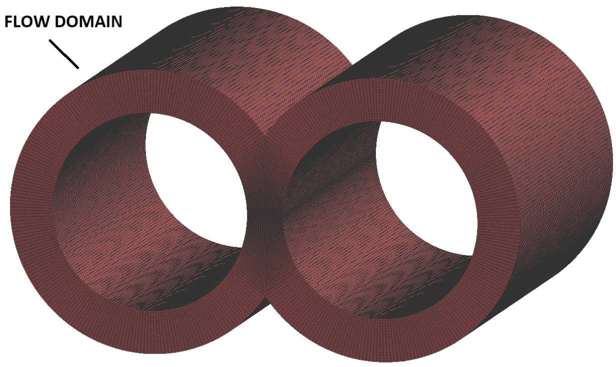

Figure 3. Mesh of fluid

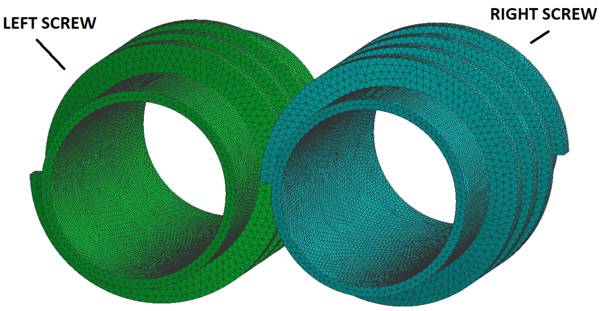

Figure 4. Mesh of twin screws for Polyflow

For the calculations in ANSYS Fluent, it was necessary to create a new geometry model for each spindle representing the area around the thread on the shaft (Figure 5). These geometries were meshed by ICEM CFD 2020 R2 program using a hexahedral element. The fluid mesh used for Polyflow had to be remeshed. The number of elements was increased in the direction of the spindle rotation axis. The Boundary Distance Based method has been set for the overset interface. This means that the overlap of the mesh is created approximately halfway between the walls. Figure 6 shows a cross section of the computational domain for ANSYS Fluent with a set and already activated overset. The total size of the computational domain reached 7,730,000 elements, with active 5,670,000 cells and 2,060,000 dead cells after overset activation.

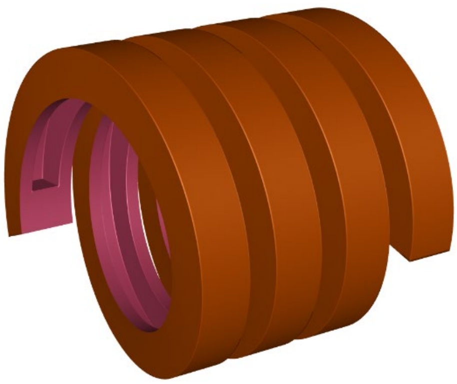

Figure 5. Geometry of left screw surrounding for component mesh

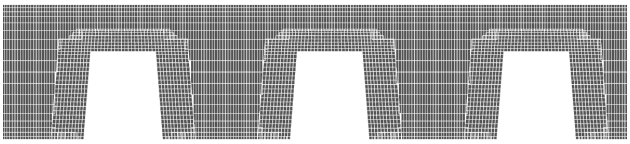

Figure 6. Section through model with overset technology activated

4.1 Boundary conditions

The choice of boundary condition is related to the required flow rate by the customer. For this required flow rate, the pump in Figure 1 was designed and manufactured by calculation.

The defined boundary conditions are the same for all CFD analyses (ANSYS Fluent, ANSYS Polyflow). The type of input boundary condition is mass flow rate with defined value $Q_m = 0.082\ kgs^{-1}$. The output boundary condition is a pressure outlet with value 0 Pa. The rotation around the spindle axes was defined, where the spindles rotate in opposite directions at an angular speed of *60 rpm*. The setup is in the Figure 7.

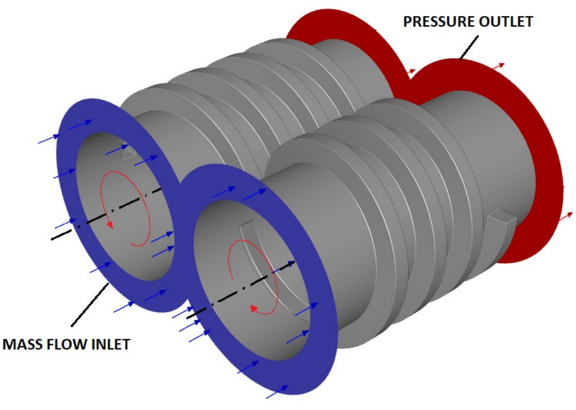

Figure 7. Boundary conditions

In the case of ANSYS Fluent, the problem is solved as transient with a time step Δt = 0.01s. The total simulation time corresponded to two complete evolutions of spindles. The optimal size of the time step Δt = 0.01s and the number of revolutions was obtained by series of CFD numerical calculations. Prior to the actual implementation of the transient calculation, the steady flow was first calculated without considering the rotation in order to obtain the initial data for transient calculation. In Polyflow, the problem is solved as steady state (pseudotransient).

Highly viscous materials, i.e. non-Newtonian fluids, are transported by means of screw pumps [Lapcik, 2020], for which it is necessary to define the corresponding functional dependence of viscosity. In this case, it is a power law definition of viscosity:

$$\eta = K\dot{\gamma}^{n-1} \tag{5}$$

Two types of non-Newtonian fluid material models (Matter A and Matter B) were defined for the calculation. As mentioned above, a power function was used to define the viscosity (Equation 5). The parameters of this function (*k* is the consistency index, *n* is flow behavior index) are shown in Table 1. The density of 1000 kgm^{-3} was used for both matters. More information you can find in [Drabkova 2020].

Parameters of viscosity model	***k* [Pas^n]**	***n* [-]**
Matter A	1381.3	0.47
Matter B	4295.8	0.1408

Table 1. Parameters of Non-newtonian power-law viscosity

The above methodology was used to verify the functionality of the model for the inlet flow boundary condition, where the model predicts the achieved inlet pressure.

5 CFD ANALYSIS EVALUATION

CFD analyses in ANSYS Fluent and ANSYS Polyflow were computed on a multi-core PC with the CFD solver set to 4 cores. The time consumption in the case of ANSYS Fluent for the material 'Matter A' was 21.5 hours, and for the model 'Matter B' was 44.5 hours. For material 'Matter A' the CFD calculation in ANSYS Polyflow took 22.5 hours, and in the case of material 'Matter B' it was 64 hours.

5.1 Comparing Fluent and Polyflow

The evaluation of the achieved results of the numerical calculations performed in ANSYS Fluent and ANSYS Polyflow is first presented in graphical form by means of filled contours of the calculated quantities (pressure and velocity) for the material 'Matter A'.

Figures 8 and 9 below show the plotted pressure contours for the 'Matter A' material. The section planes for contours pass through the rotation axis of the left spindle and are oriented in the vertical and horizontal directions. In the case of the results obtained in ANSYS Fluent (Figure 8), the location with the highest pressure is at the beginning of the left spindle (LS) thread. While in ANSYS Polyflow, the location of highest pressure is on the right spindle. The maximum pressure achieved differs by 53 kPa.

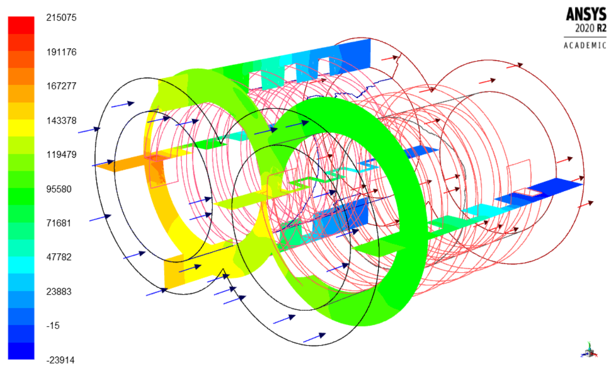

Figure 8. Contours of pressure distribution [Pa] in ANSYS Fluent for material model 'Matter A'

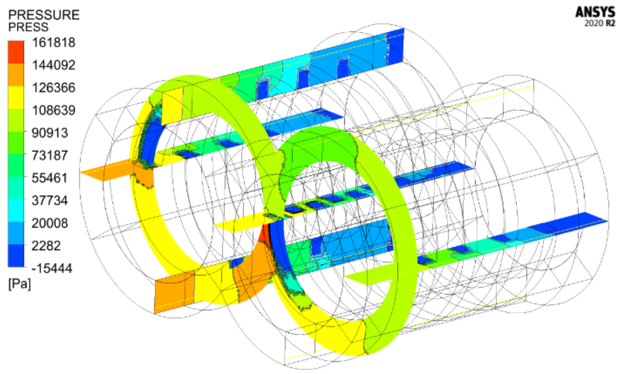

Figure 9. Contours of pressure distribution [Pa] in ANSYS Polyflow for material model 'Matter A'

The results of the pressure distribution show a very good match in terms of the pressure distribution behaviour in individual section planes through the computational domain.

The contours of the velocity magnitude are plotted in the Figure 10 and 11. In both cases, the location with the highest velocity is in the gap between the spindles, i.e. in the space between the teeth of the threads. In the case of ANSYS Fluent, the maximal achieved speed is equal to 0.275 ms^{-1}, while in the case of ANSYS Polyflow it is 0.333 ms^{-1}. The contours of the velocity magnitudes differ slightly from each other, since only thread rotation is considered in the case of the computational model for ANSYS Fluent. While shaft rotation is not considered and therefore there is a slight distortion of the results obtained by ANSYS Fluent to the real situation.

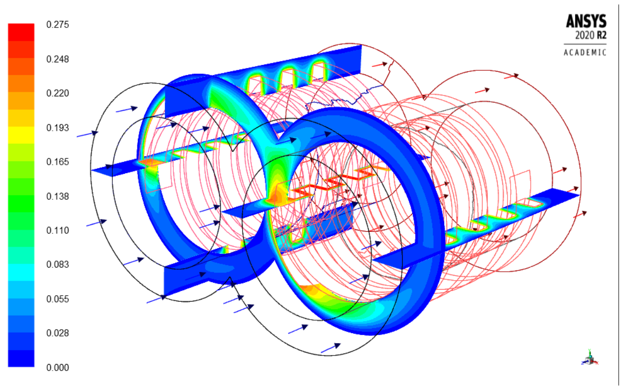

Figure 10. Contours of velocity [ms^{-1}] in ANSYS Fluent for material model 'Matter A'

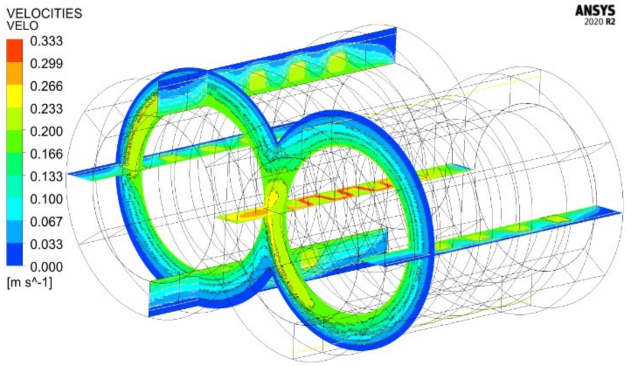

Figure 11. Contours of velocity [ms^{-1}] in ANSYS Polyflow for material model 'Matter A'

5.2 *Pressure Distribution in axial direction*

For the purpose of better illustration and to compare the two software, the pressure distribution was evaluated along the length of the spindles in the gaps by using line segments for both material models. For this evaluation it was necessary to define and create 8 line segments in the model on which the pressure distribution was plotted. These line segments are located between the threads and the stator wall. They are named according to the cardinal directions and the spindle near which they are located (LS - Left Screw, RS - Right Screw) as can be seen in the Figure 12. In the plots, the pressure is plotted in the axial direction, where coordinate 0 m is the inlet flow boundary condition and coordinate -0.12 m is the outlet pressure boundary condition.

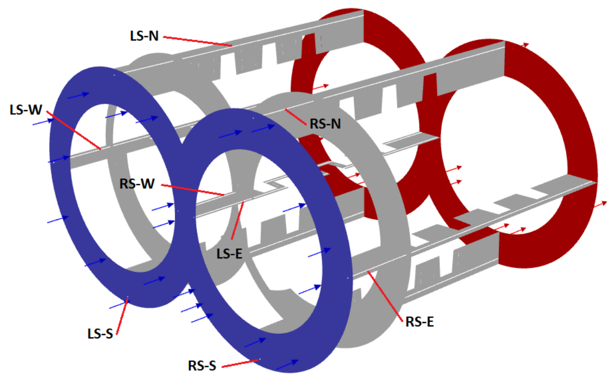

Figure 12. Positions of the line segments for the evaluation of the pressure distribution

The first pair of graphs (Figure 13 and 14) show the pressure distribution for the material 'Matter A'. As mentioned in the contour evaluation, in the case of ANSYS Fluent, a higher pressure was achieved at the first spindle, this can be seen on the LS-W line segment. In terms of the trend and character of the pressure distribution behaviour, it can be noted that there is a very good match between the results obtained with ANSYS Fluent and ANSYS Polyflow.

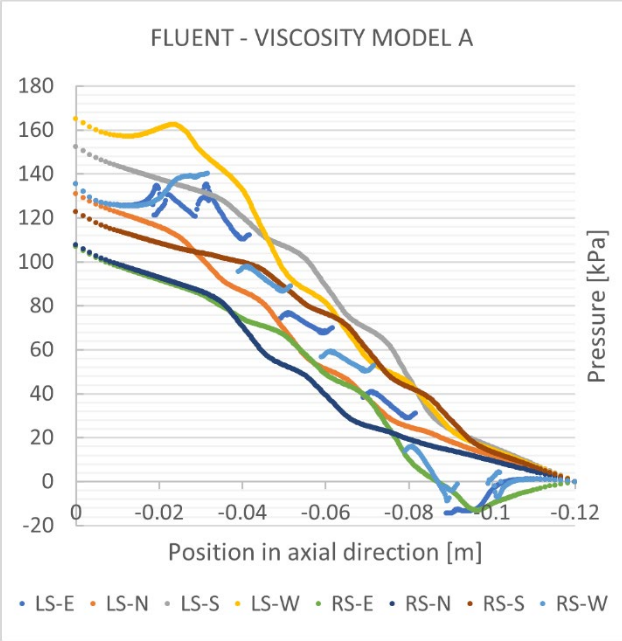

Figure 13. ANSYS Fluent axial direction pressure distribution on the line segments for material 'Matter A'

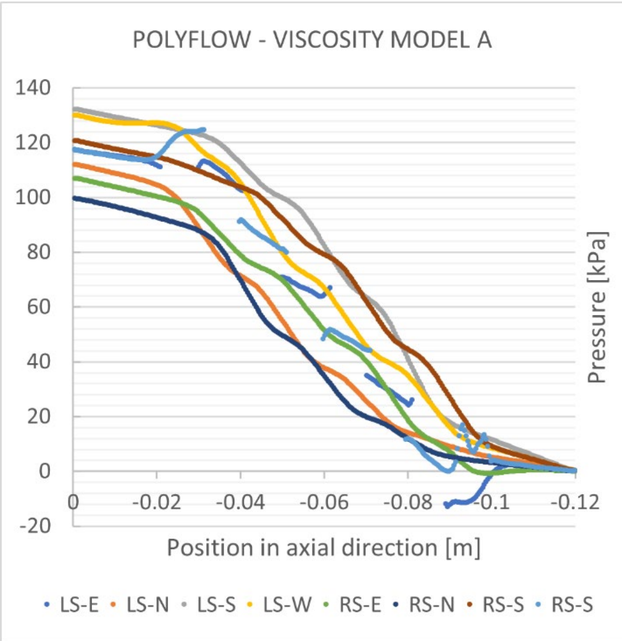

Figure 14. ANSYS Polyflow axial direction pressure distribution on the line segments for material 'Matter A'

Similar conclusions in terms of the trend and character of the pressure distribution along the length in the gaps of the spindles can be stated in the case of the results obtained with ANSYS Fluent and ANSYS Polyflow (Figure 15 and 16) for material 'Matter B'.

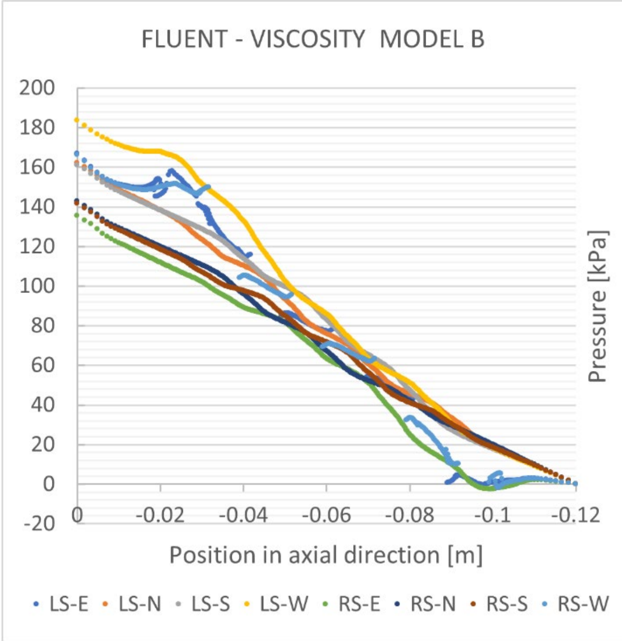

Figure 15. ANSYS Fluent axial direction pressure distribution on the line segments for material 'Matter B'

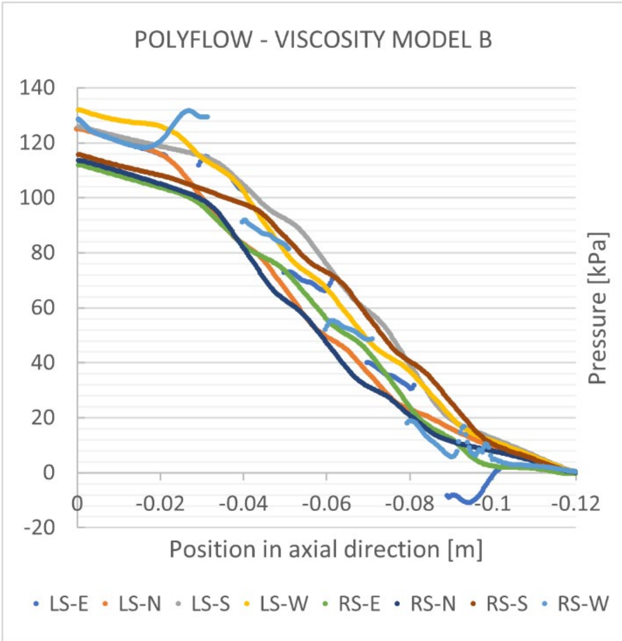

Figure 16. ANSYS Polyflow axial direction pressure distribution on the line segments for material 'Matter B'

5.3 Mesh size and quality influence in Polyflow

The effect of the size and quality of the fluid mesh on the results in ANSYS Polyflow software was also tested. In total, three different computational meshes were created for the flow domain ('eight' shape mesh). This resulted in computational domains of 700,000 elements, 1,500,000 elements and 2,000,000 elements. For a good visual representation of the results obtained, only 2 line segments in the axial direction (LS-N and RS-S) were selected. The results are shown in the Figure 17.

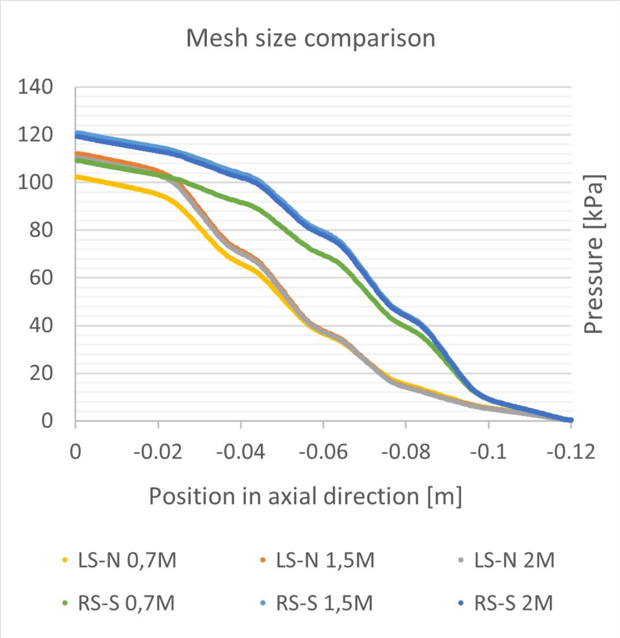

Figure 17. ANSYS Polyflow axial direction pressure distribution on the line segments for different size computational domains

The results show that the network with 0.7 million elements differs significantly from the other two networks. The difference in the results of the 1.5 million and 2 million network is within 1.5%. Based on these conclusions, all calculations in ANSYS Polyflow were performed on the computational network with 1.5 million elements.

6 CONCLUSIONS

Two computational models were created for ANSYS Fluent and ANSYS Polyflow software based on the real geometry for the screw pump. The geometry was modified, specifically increasing the spindle gap to 2.5 mm to reduce computational requirements and speed up calculations, as this is the first phase of the newly proposed screw pump design methodology using overset. For this technology it was necessary to prepare component meshes around the threads. This process proved to be time consuming compared to ANSYS Polyflow where the spindle geometry was meshed directly. In addition to the need for creation of new meshes for overset, there is also a higher requirement for this technology to ensure the quality of the meshes and their proper overlap with each other for proper data interpolation. Thus, the overall preparation process for ANSYS Fluent calculations is significantly more demanding. This disadvantage will be eliminated soon by the presence of new tools for creating overset meshes in the latest version of ANSYS Fluent 2021 R2 with meshing mode.

Based on previous research/work, two material models with viscosity defined by a power function were developed for the calculations. After defining the boundary conditions of the CFD calculations (mass flow intlet with a value of 0.082 kgs^{-1}, pressure outlet with a value of 0 Pa and counter-rotation of the spindles with a speed of 60 rpm), the calculations were then performed. The total computational domain size for ANSYS Fluent reached 7,730,000 cells, leaving 5,670,000 active cells and 2,060,000 dead cells after the overset was activated. The computations took 21.5 hours for the 'Matter A' and 44.5 hours for the 'Matter B' material model. For ANSYS Polyflow, the resulting computational domain had 1,500,000 cells and took 22.5 hours to compute for the 'Matter A' and 64 hours for the 'Matter B' material model.

The results obtained by both software (ANSYS Fluent, ANSYS Polyflow) show a very good match in terms of the pressure distribution behaviour in individual section planes through the computational domain. Therefore, further development of this procedure using overset in spindle design can be considered. Results differ slightly in the magnitude of the observed variables. The difference in the resulting values is probably due to the computational model for ANSYS Fluent. In this case, it is an initial design of the computational model, where the rotation is not applied directly to the shaft, but only to the thread, therefore there is a distortion to reality. In the next version of the computational model, a separate component mesh for the shafts is already planned, to which rotation will be applied too. The effect of the inlet and outlet pressure boundary condition on the methodology will also be tested.

With the new tools in ANSYS version 2021 R2 for creating oveset meshes and the possibility of remeshing overset meshes for narrow gaps during the calculation, there will be an overall reduction in elements and faster solution.

ACKNOWLEDGMENTS

The work presented in this paper was supported by a grant SGS 'Control of Fluid Systems and their Parameters Measurement' SP2021/85 and grant FV40105 'Development and production of high-pressure spindle pump prototype for pumping high-viscosity masses'.

REFERENCES

[ANSYS Fluent, 2020] *ANSYS Fluent, Release 2020 R2: User's Guide [online]. Help System, User's guides, ANSYS, Inc, 2021* [2021-09-29]. Available from https://ansyshelp.ansys.com/.

[ANSYS Polyflow, 2020] *ANSYS Polyflow, Release 2020 R2: User's Guide [online]. Help System, User's guides, ANSYS, Inc, 2021* [2021-09-29]. Available3 from https://ansyshelp.ansys.com/.

[Drabkova 2020] Drabkova, S. Bojko, M. Vesely, J. Zapletal, R. Experimental Investigation and Numerical Modelling of Flow Characteristics of Non-Rotating Twin Screw Model In: MATEC Web of Conferences 328 (AEaNMiFMaE-2020), Zilina, 2020. eISSN: 2261-236X.

[Lapcik, 2020] Lapcik, L. Vsina, M. Lapcikova, B. Burecek, A. Hruzik, L. Effect of rapeseed oil on the rheological, mechanical and thermal properties of plastic lubricants. MECHANICS OF TIME-DEPENDENT MATERIALS, 2020. ISSN 1385-2000.

[Lewandowski, 2017] Lewndowski, A. Wilczynski, K. J. Computer modeling for polymer processing co-rotating twin screw extrusion-nonconventional screw configurations. MECHANIK, 2017, Vol. 90, No.4., pp. 282-287. ISSN 0032-2725.

[Ryazantsev, 2010] Ryazantsev, V. M. Plyasov, V. V. Determining the Forces on the Screws in Two- and Three-Bearing Two-Screw Pumps. Rusian Engineering Research, 2010, Vol.30, No.9., pp. 877-885. ISSN 1068-798X.

[Tagliavini, 2016] Tagliavini, G. Solari, F. Montanari, R. In: Proceedings of the International Food Operations and Processing Simulation Workshop, Bruzzone, Longo, Piera, September, 2016. Universita DI Genova, pp. 32-38. ISBN 978-88-97999-75-1.

[Wilczynski, 2016] Wilczynski, K. J. Nastaj, A. Modelowanie procesu wytłaczania jednoslimakowego mieszanin polimerow z zastosowaniem slimakow niekonwencjonalnych i dozowanego zasilania wytłaczarki. POLIMERY, 2016, Vol.61, No.5., pp. 357-362. ISSN 0032-2725.

[Xu 2018] Xu B, Liu Y, He L, Chen J, Turng L-S. Numerical study of mixing dynamics inside the novel elements of a corotating nontwin screw extruder. Adv Polym Technol. 2018; 37:2478–2496. https://doi.org/10.1002/adv.21923.

CONTACTS:

Doc. Ing. Marian Bojko, Ph.D
VSB-Technical University of Ostrava/Faculty of Mechanical Engineering/Department of Hydromechanics and Hydraulic Equipment
VSB-TU Ostrava, Faculty of Mechanical Engineering, 17. listopadu 2172/15, 708 00
+420 596 994 385, marian.bojko@vsb.cz, www.vsb.cz