The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM)
ISSN: 2602-3199

The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 2023

Volume 26, Pages 462-473

IConTES 2023: International Conference on Technology, Engineering and Science

Investigation of the Flow Performance of the Diverter Cover in the Dishwasher

Vasıf Can Yildiran
Haier Europe- Renta Elektrikli Ev Aletleri San. Tic. Ltd. Sti.

Fazıl Erinc Yavuz
Haier Europe- Renta Elektrikli Ev Aletleri San. Tic. Ltd. Sti.

Sebastian George Colleoni
Haier Europe

Abstract: Competition remains fierce within the dishwasher industry, mirroring the ongoing rivalry in all household appliance sectors. The influence of fresh regulations and advancements in technology has spurred a multitude of research endeavors. In the context of household dishwashers, it is widely recognized that the primary focal points are centered on washing efficiency, drying capabilities, water consumption, and energy efficiency. The effectiveness of the hydraulic system within the machine plays a pivotal role in determining its washing prowess, water usage, and energy performance. This research focuses on the examination of the geometry of the diverter element's cover within the hydraulic system of a dishwasher. The diverter and its associated cover, crucial components of the hydraulic circuit, underwent a comprehensive analysis utilizing Computational Fluid Dynamics (CFD) software. Subsequent to the analysis, efforts were made to enhance their performance, encompassing both experimental and numerical approaches. As a result of optimizing the diverter cover's geometry, a notable reduction of 26 ml in water volume within the reservoir was achieved, along with a substantial improvement in pressure loss, amounting to 20 mbar. Subsequent tests conducted with the improved design showcased a remarkable increase in flow rate, measuring 8.1% at 2400 pump revolutions per minute and 8.9% at 2800 pump revolutions per minute.

Keywords: Circulation pump, Computational fluid dynamics (CFD), Dishwasher, Hydraulic

Introduction

Dishwashers have become indispensable appliances in our homes, serving various purposes in both domestic and industrial settings. These machines, available in various types, are designed to efficiently wash and dry dirty dishes while minimizing water, energy, and noise consumption. The concept of dishwashers with continuous water spraying systems was first introduced with William Howard Livens' household-friendly dishwasher in 1924. The modern dishwasher, as we know it today, evolved towards the end of the 1970s (Livens, 1924). Since then, ongoing efforts have been dedicated to the development of more efficient dishwashers in response to resource constraints and evolving regulations.

The effectiveness of a dishwasher's performance hinges significantly on its hydraulic design. Enhancing this design directly translates to improved machine efficiency. It is imperative to minimize pressure losses within the hydraulic circuit components and throughout the circuit itself, as well as to conduct a comprehensive analysis of the flow within the hydraulic system.

The hydraulic system within a dishwasher is responsible for delivering pressurized water to the dishes with the least energy consumption and minimizing pressure loss. This system comprises critical components, including a circulation pump, diverter, sump, delivery tube, and spray arms. This particular study examined the diverter cover, an important component of the hydraulic system in the dishwasher. The investigations encompass not only diverter models, which are electromechanical components controlling the distribution of flow to the spray arms but also a study of a fixed model incorporated into the diverter cover. This region, referred to as the diverter cover, facilitates the distribution of water to both upper and lower spray arms in models without diverters. In models equipped with a diverter mechanism, the distribution of water to the spray arms operates based on the position of the diverter, which changes according to an algorithm. In both operating scenarios, the diverter region experiences high flow velocities and pressure losses.

The study encompasses both numerical and experimental methodologies. Computational Fluid Dynamics (CFD) analyses were conducted using Ansys Fluent software. Additionally, the experimental phase involved the assessment of finalized designs resulting from the CFD analysis in Haier Europe Washer R&D laboratories. Energy performance tests for dishwashers, on the other hand, were carried out in compliance with the EN 50242 standard at Haier Europe Energy Performance laboratories.

Hydraulic System of Dishwasher

The hydraulic system comprises several key components, including the circulation pump, sump, diverter, delivery tube, and spray arms. These components are strategically positioned within the dishwasher, as depicted in the CAD render below. It's important to note that the diverter location, a vital part of the hydraulic system, is not visible in this visual representation.

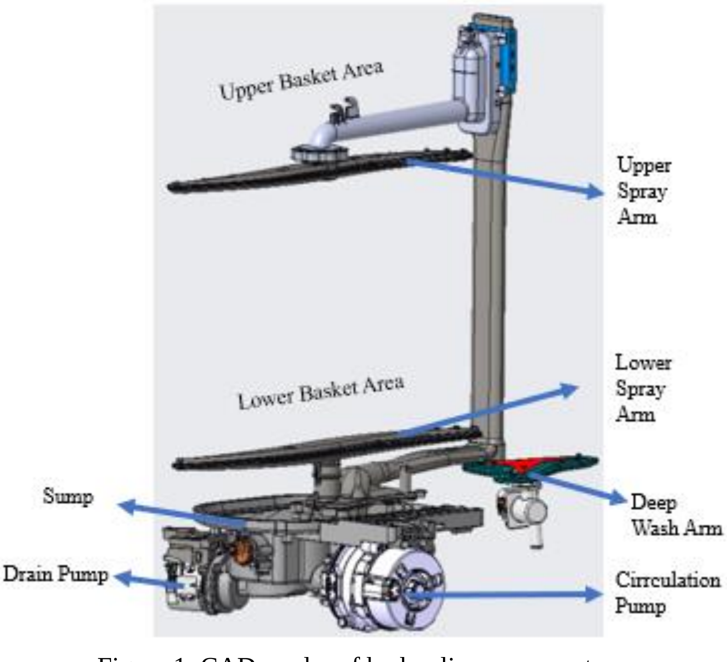

Figure 1. CAD render of hydraulic components

The hydraulic system's water cycle operates in the following sequence: Initially, water is pressurized by the circulation pump and subsequently directed to the diverter. Depending on the selected wash function, the diverter adjusts its position to guide the flow appropriately, or in models without diverters, all arms operate directly. The pressurized water then travels through connection points to reach the spray arms, from where it is expelled toward the dishes through nozzles on the spray arms.

Afterward, the water, having passed over the dishes, collects on the bottom surface of the machine and undergoes filtration before descending into the sump. From the sump, the water proceeds through the circulation pump channel, eventually reaching the impeller of the circulation pump. Here, it is primed to be pumped back to the spray arms, thereby continuing this cycle until the washing process is completed or until the dirty water is drained to make way for clean water replacement.

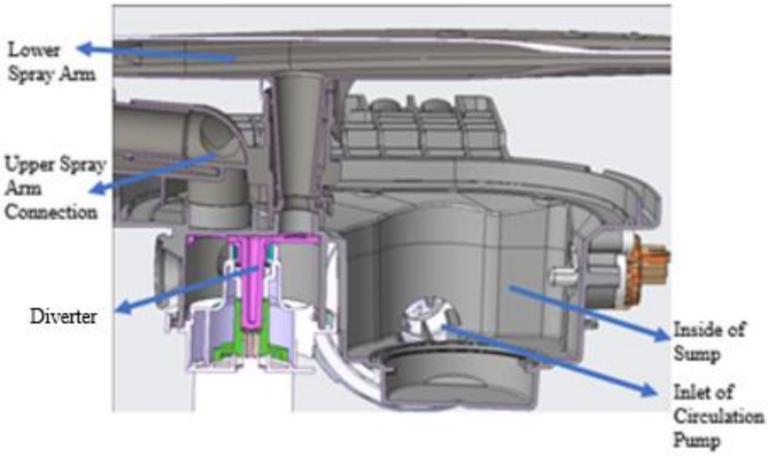

Figure 2. CAD rander of hydraulic components of sump area section

Aim of the Study for Diverter Cover

This study is related to the diverter region in the dishwasher, which follows the circulation pump and is responsible for distributing the flow to the spray arms. In this region, there is either an electromechanical component controlled by an algorithm that directs water to the spray arms based on the diverter's position, or there is a plastic geometry incorporated directly into the diverter cover, which distributes the flow to the spray arms. In the study, the aim was to minimize pressure losses in this region and analyze the flow rates going to the spray arms in non-diverter situations. Therefore, the analyses in the relevant area were examined, and appropriate geometry studies were conducted.

Method

Theory

Pipe Flow

Within a dishwasher's hydraulic system, the liquid follows a path through enclosed conduits before it ultimately reaches the tub. This particular flow, contingent on the cross-sectional shape of the conduits, is often classified as either pipe flow or duct flow (Munson et al., 2012). Figure 3 provides a visual representation of a standard pipe system, showcasing several of the typical components that constitute such a system.

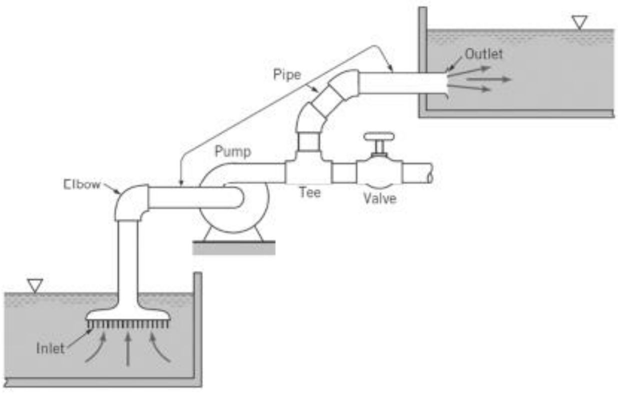

Figure 3. A pipe system and some hydraulics components flow (Munson et al., 2012)

Flow Regime in Pipe Flow

In the realm of fluid mechanics, this type of flow is categorized into two distinct regimes known as Laminar and Turbulent flow. These classifications were established based on experiments conducted by the British scientist

Osborne Reynolds (1842-1912). Laminar flow refers to the smooth movement of fluid in parallel layers, with no intermixing or interference between these layers (Batchelor, 1967). In contrast, turbulent flow is characterized by chaotic and irregular behavior, including significant lateral mixing of the fluid layers.

This classification primarily pertains to flows inside pipes, where both laminar and turbulent flows can manifest. Within the context of pipe flows, the Reynolds number holds immense significance as a crucial dimensionless quantity. For a pipe with a diameter represented as "D" and an average velocity denoted as "V," the Reynolds number can be expressed as follows:

$$Re = \frac{\rho V D}{\mu} \qquad (1)$$

In the context of the Reynolds number equation, where ρ represents fluid density and μ represents dynamic viscosity, it's important to note that the precise range at which the transition from laminar to turbulent pipe flows occurs cannot be precisely defined. This transition can take place at different Reynolds numbers, influenced by factors such as pipe vibrations and the roughness of the entrance region.

For practical engineering applications, the following values are commonly utilized: In a round pipe, flow is typically categorized as laminar if the Reynolds number is approximately below 2100 and as turbulent if it is approximately above 4000 (Munson et al., 2012). Within the intermediate range encompassed between these two thresholds, the flow is considered transitional and is highly susceptible to disturbances. Within a dishwasher's hydraulic system, the Reynolds number falls within the range of 3000 to 20000 for a pump speed of 1800 rpm and between 8000 to 30000 for a pump speed of 2800 rpm, predominantly resulting in turbulent flow. Therefore, for the purposes of this study, since the pump speeds are higher than 1800 rpm, the flow can be considered turbulent.

Energy Considerations in Pipe Flow

One of the fundamental principles in physics is the conservation of energy. Consequently, it is of paramount importance to analyze pipe flows from an energy perspective. When dealing with an incompressible fluid flowing inside a pipe, assuming steady flow along a streamline and neglecting the effects of friction (inviscid flow), the well-established Bernoulli equation can be applied.

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = constant \qquad (2)$$

Indeed, the Bernoulli equation provides valuable insights into the factors that contribute to pressure changes in the case of inviscid flow along a streamline. It highlights how changes in certain parameters affect the pressure within the fluid. Specifically:

A decrease in the flow area (A) leads to an increase in fluid velocity (V), which is associated with a decrease in pressure (P). Conversely, if the flow area expands, or if the pipe slopes downward, the velocity of the fluid decreases, resulting in an increase in pressure.

This principle, elucidated by the Bernoulli equation, helps us understand how changes in flow velocity, pipe geometry, and elevation influence the pressure distribution within a fluid system (Pritchard & Mitchell, 2016).
Nevertheless, flows within pipes and ducts are subject to substantial friction and often exhibit turbulence, rendering the Bernoulli equation inapplicable. Essentially, the influence of friction results in a continuous decrease in the value of the Bernoulli constant as defined in Equation (2), leading to a dissipation of mechanical energy. Consequently, it becomes necessary to replace the Bernoulli equation with an energy equation that accounts for the effects of friction (Pritchard & Mitchell, 2016). In pursuit of this, we examine the energy equation for incompressible, steady flow between two points, as provided below.

$$\frac{p_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{l_T} \qquad (3)$$

The coefficients α_1 and α_2 are used to account for kinetic energy losses due to non-uniform velocity profiles across the pipe. When dealing with a perfectly uniform velocity profile, α equals 1. However, for any non-uniform velocity profile, α is less than 1. In practice, for high Reynolds numbers, α is often very close to unity because the change in kinetic energy is usually small compared to the dominant terms in the energy equation. Therefore, the approximation α ≈ 1 is commonly employed in calculations involving pipe flow (Pritchard & Mitchell, 2016).

The term h_{l_T} represents the head loss, which considers energy dissipation resulting from viscous effects present throughout the fluid in the pipe flow (Munson et al., 2012). This head loss accounts for the energy that is lost as the fluid flows through the pipe due to friction and other factors, contributing to a decrease in the overall energy of the fluid as it moves along the pipeline.

To calculate the pressure difference between any two points within a fluid system using Equation (3), it is crucial to accurately determine the head loss. The head loss accounts for energy dissipation due to various factors such as friction, non-uniform velocity profiles, and other influences that lead to a decrease in the total energy of the fluid as it traverses the system. Accurate estimation of the head loss is essential for understanding pressure changes and optimizing the performance of the fluid system. The head loss term, h_{l_T}, is often known as the total head loss, and it comprises two main components:

Major losses, denoted as h_l, which are primarily attributed to frictional effects occurring in fully developed flow within constant-area tubes. These losses are related to the resistance that the fluid encounters as it moves through the pipe due to friction with the pipe walls.

Minor losses, represented as h_{l_m}, which arise from various factors such as entrances, fittings, changes in area, and other elements within the fluid system. These minor losses are associated with deviations from ideal flow conditions, including abrupt changes in flow direction and the presence of components like valves and elbows (Pritchard & Mitchell, 2016). Both major and minor losses contribute to the overall head loss in a fluid system, and their accurate consideration is vital for understanding and managing pressure changes in the system.

CFD Methodology

Model Preparation

The initial phase of the CFD study involves model preparation. To begin, the specific region of interest is identified using the corresponding part code within a CAD program and then saved in .stp format onto the computer. Following this step, the file is imported into SpaceClaim software. Here, a thorough review of the geometric details on the part is conducted, and any elements found to be non-impactful on our CFD analysis results but potentially adding complexity to the solution mesh are systematically removed to optimize the model.

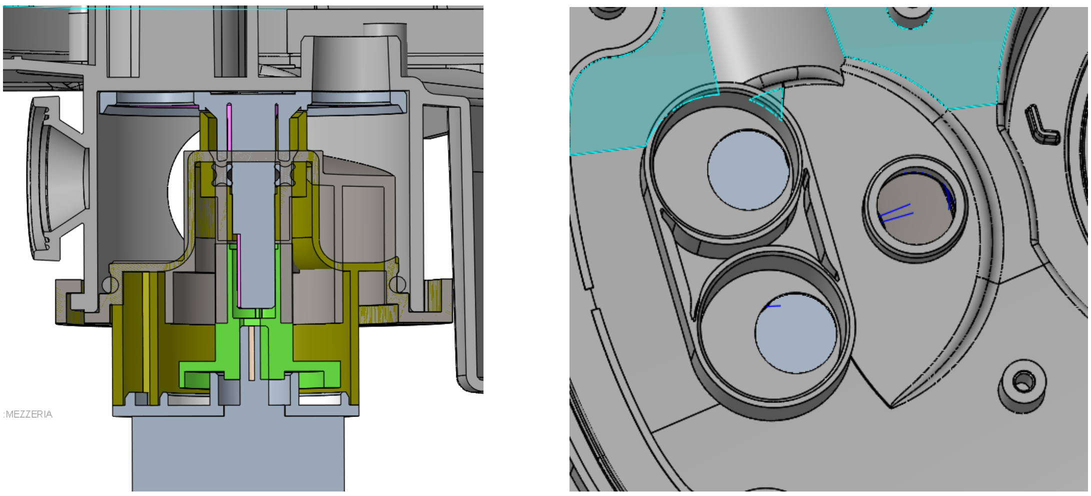

Figure 4. CAD model of diverter area section and top view of diverter spray arm connection

Diverter and non-diverter systems vary depending on the dishwasher model. In dishwashers with a diverter chamber, it is present to distribute water to the spray arms in models without diverters. In this study, the geometries of non-diverter systems in the diverter chamber have been examined.

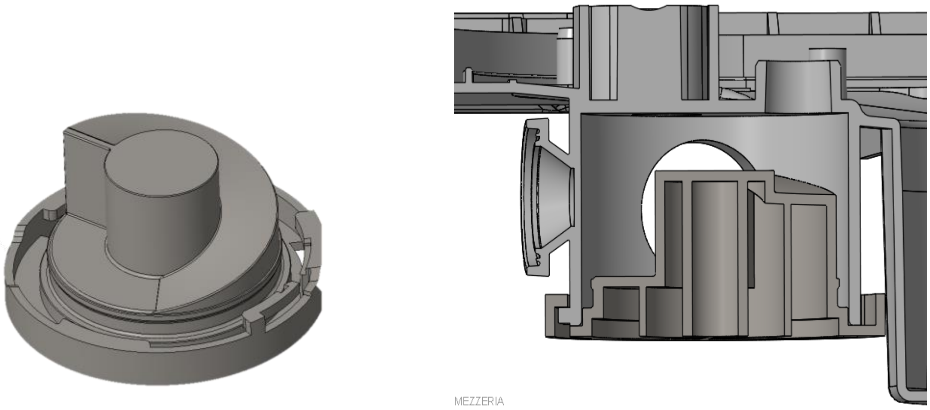

Figure 5. CAD model of current diverter cap and section of diverter area with current cap

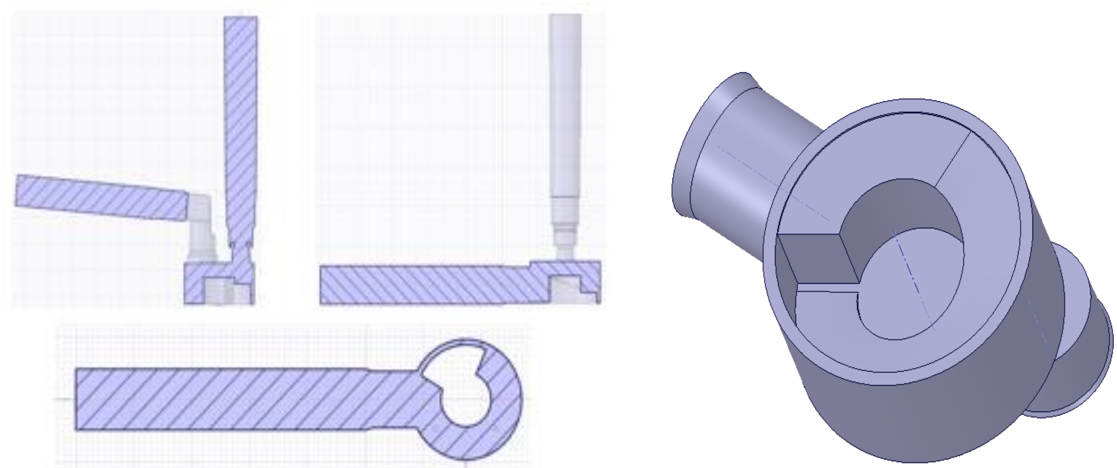

Figure 6. Some sections of full model fluid volume and bottom view of current cap diverter area

The necessary flow volume has been obtained for conducting fluid dynamics analysis. Inside this volume, a solution mesh will be created using Ansys Mesh software.

Mesh Preparation

Ansys Mesh software has been chosen for meshing the flow volumes. During the mesh creation process, finer elements have been implemented in areas where flow separation is anticipated. Additionally, a boundary layer mesh has been applied to solid surfaces to capture solid-liquid interactions. Before conducting CFD analyses, a mesh independence study has been carried out. The goal of this study is to ensure that the analyses are not influenced by the mesh cell count and sizes. The mesh independence study was conducted based on the variation of velocity values read from a specific of diverter inlet. At the end of the study, analyses were carried out with a mesh count of 1 million.

Table 1. Mesh independency

Study No	1	2	3	4	5
Mesh Number	510 k	820 k	1 M	1.2 M	1.4 M
Velocity for Control Point [m/s]	1,6	1,66	1,9	1,91	1,91
Skewness	0,87	0,86	0,85	0,85	0,83

The mesh count has been defined for the baseline, and a mesh independence study has been completed. The mesh count for each geometry varies according to geometric details, but there was no need to conduct another mesh independence study.

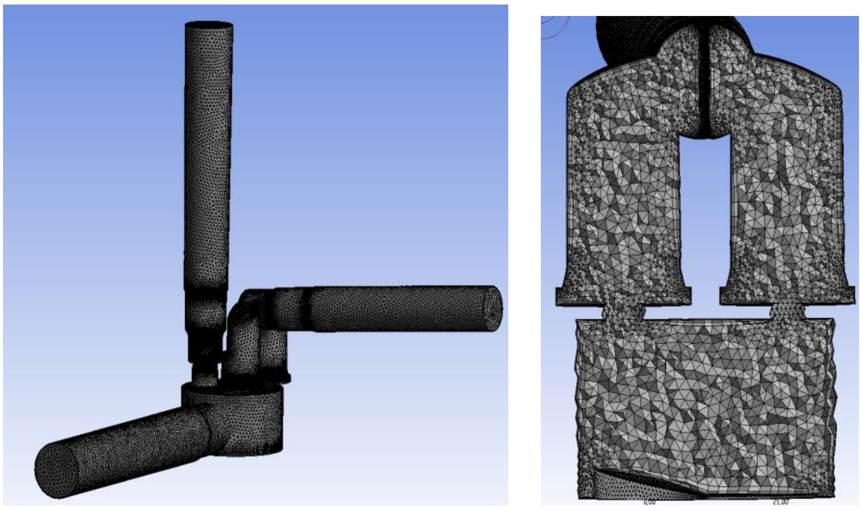

Figure 6. Mesh of full body and full body section

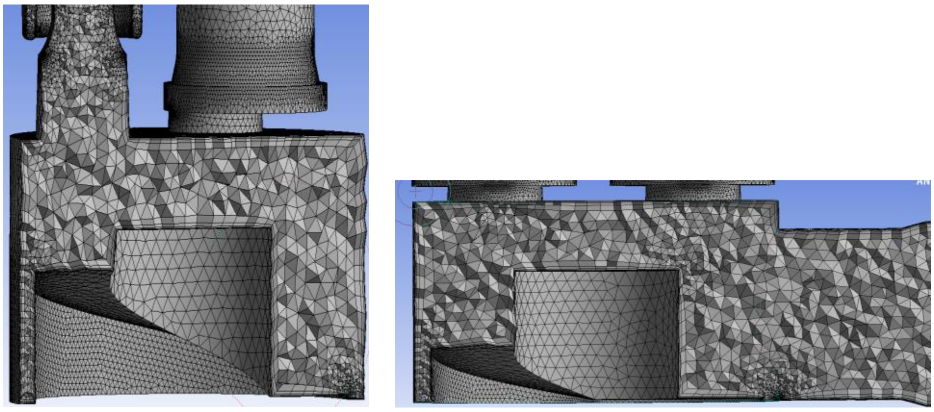

Figure 7. Mesh details for some sections

CFD Model and Studies

Design optimization studies (CFD) have become increasingly prevalent in numerous engineering projects. These studies help achieve cost savings in terms of prototypes and time, allowing for the testing of multiple designs within a short timeframe. In this study, Ansys Fluent software has been utilized for the CFD analyses. After creating the solution mesh using Ansys Mesh, boundary conditions for the CFD analysis were established using Fluent, as detailed in the Ansys User Guide.

The study focuses on determining the boundary conditions for the model in use. The numerical model has been adopted under the assumptions of incompressible, time-independent flow, and without energy equations. The fluid within the model is single-phase, using water as the working fluid. Atmospheric pressure has been set as the inlet condition for the diverter inlet channel, while the model's outlets are defined by the flow rate provided by the pump. The focus of this study is the pressure losses caused by the diverter models under investigation. Therefore, experimental data for the flow rates in the lower and upper spray arms have been used, and the same approach has been applied to all models. The boundary condition at the inlet of the flow is defined as the pressure input.

Table 2. Boundary conditions

Input Definition	Values
Pressure Inlet	101325 Pa
Volume Flow Rate Upper	28.5 lt/min
Volume Flow Rate Lower	26.5 lt/min
Density	992.2 kg/m^3
Dynamic Viscosity	0.0006533 Pa.s

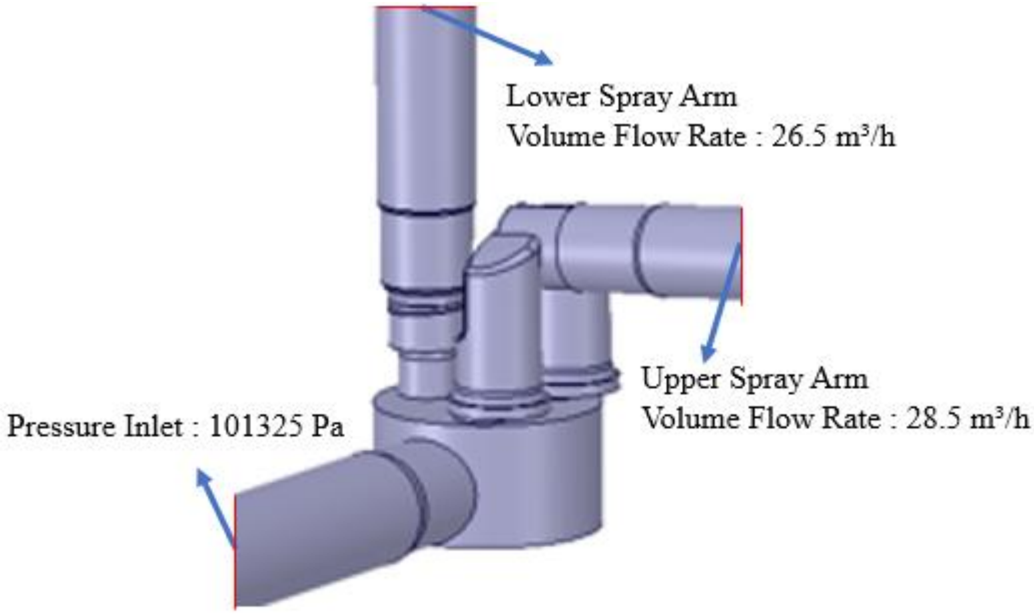

Figure 8. Boundary conditions on CFD model

At the outset of the research, the flow characteristics of the model were identified as turbulent using the Reynolds number relationship. The pressure-velocity coupling in the Navier-Stokes equations was resolved using the Coupled solution method (Wilcox, 1998). The equations for continuity and momentum employed in the solution are provided below.

$$\left(\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z}\right) = 0 \qquad (3)$$

$$\rho U\left(\frac{\partial U}{\partial x} + \frac{\partial V}{\partial x} + \frac{\partial W}{\partial x}\right) = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 U}{\partial x^2} + \frac{\partial^2 U}{\partial y^2} + \frac{\partial^2 U}{\partial z^2}\right) - \mu\left(\frac{\partial \overline{u'^2}}{\partial x^2} + \frac{\partial \overline{u'v'}}{\partial x \partial y} + \frac{\partial \overline{u'w'}}{\partial x \partial z}\right) \qquad (5)$$

$$\rho V\left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial y} + \frac{\partial W}{\partial y}\right) = -\frac{\partial P}{\partial y} + \mu\left(\frac{\partial^2 V}{\partial x^2} + \frac{\partial^2 V}{\partial y^2} + \frac{\partial^2 V}{\partial z^2}\right) - \mu\left(\frac{\partial \overline{u'v'}}{\partial x \partial y} + \frac{\partial \overline{v'^2}}{\partial y^2} + \frac{\partial \overline{v'w'}}{\partial y \partial z}\right) \qquad (6)$$

$$\rho W\left(\frac{\partial U}{\partial z} + \frac{\partial V}{\partial z} + \frac{\partial W}{\partial z}\right) = -\frac{\partial P}{\partial z} + \mu\left(\frac{\partial^2 W}{\partial x^2} + \frac{\partial^2 W}{\partial y^2} + \frac{\partial^2 W}{\partial z^2}\right) - \mu\left(\frac{\partial \overline{u'w'}}{\partial x \partial z} + \frac{\partial \overline{v'w'}}{\partial y \partial z} + \frac{\partial \overline{w'^2}}{\partial z^2}\right) \qquad (7)$$

A range of turbulence methods and models are utilized for conducting Heat and Mass Diffusion (HAD) analyses. While Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) approaches are popular for fluid dynamics applications, this study has chosen the Reynolds-Averaged Navier-Stokes (RANS) approach, which is also widely recommended in engineering literature. The RANS approach is favored over other techniques more frequently due to its cost-effectiveness and applicability. Within the Ansys Fluent software used for HAD simulations, the k-ω SST (Shear Stress Transport) turbulence model has been selected from the available RANS options (Menter, 1994).

$$\rho\frac{\partial k}{\partial t} + \rho u_i \frac{\partial k}{\partial x_j} = P - \beta^* \rho k \omega + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_k \mu_T)\frac{\partial_k}{\partial x_j}\right] \qquad (8)$$

$$\rho\frac{\partial \omega}{\partial t} + \rho u_i \frac{\partial \omega}{\partial x_j} = \frac{\gamma\rho}{\mu_T} P - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_\omega \mu_T)\frac{\partial \omega}{\partial x_j}\right] + 2\rho(1 - F_1)\frac{\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \qquad (9)$$

Post Process

The CFD analyses were initially conducted for the diverter cap without any specific geometry. Subsequently, several models aimed at minimizing pressure losses were created. Analyses of CFD for all these models were completed, and pressure losses at the same control points were examined. Another topic is that volume reduction of diverter area to decrease water consumption.

The results of the currently employed model, Model A, have been examined. Other models have been developed to decrease pressure losses. The baseline model, on the other hand, was modeled solely to understand how results could be obtained when there is no figure on diverter cap. The baseline, Model A which is current production and other models are presented as shown in the following Figure 9.

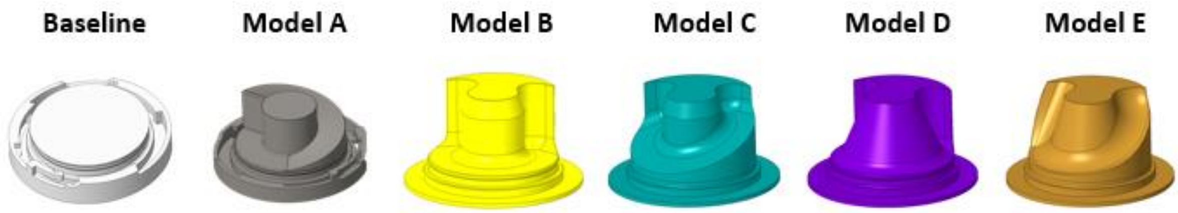

Figure 9. Diverter cap models

In order to examine the velocity distribution within the flow, velocity vectors, streamlines, and contours have been obtained from various cross-sections as shown in the following Figure 10 and 11.

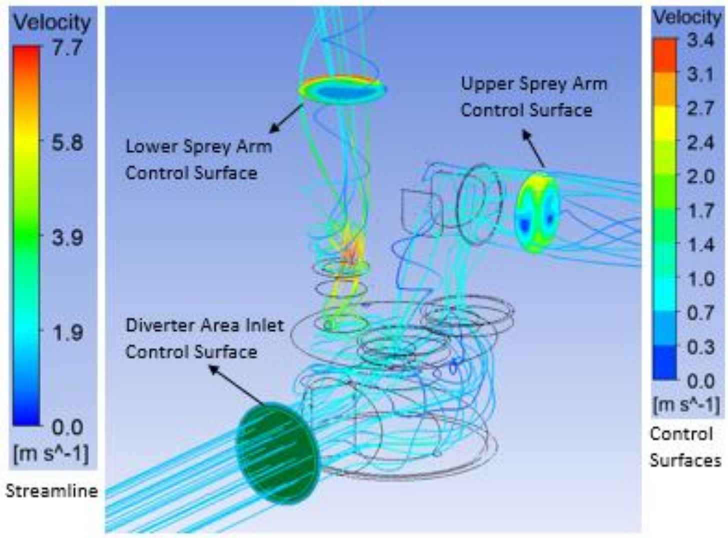

Figure 10. Streamlines and velocity distribution on control surfaces

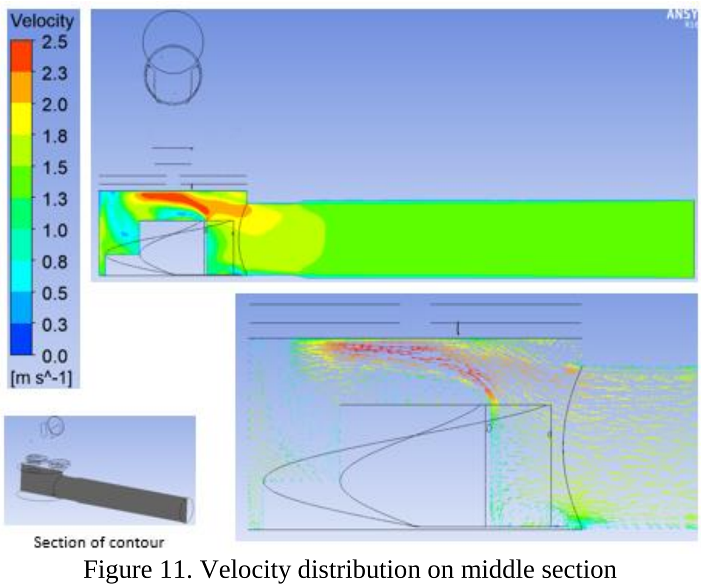

Figure 11. Velocity distribution on middle section

One of the main objectives of the study is to examine pressure losses. To understand pressure losses, pressures on the relevant control surfaces have been measured as shown in the following Figure 12.

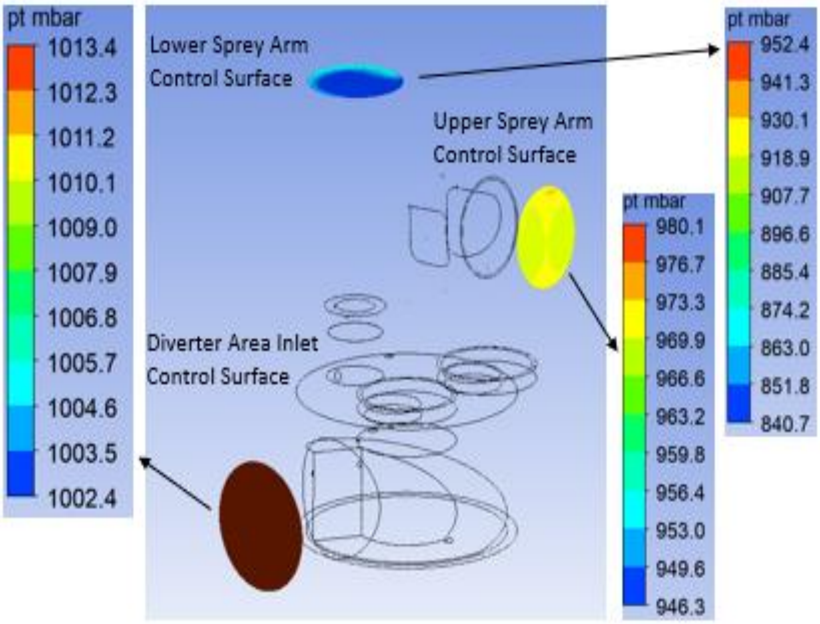

Figure 12. Total pressure value on control surfaces

The CFD results of Model A, which is currently used in the production, have been examined. Based on these results, it was observed that the relevant geometry induces a swirl motion in the flow, and it was determined that this swirl flow causes pressure losses. In the other models, the goal is to reduce these pressure losses and, at the same time, minimize the volume of the diverter chamber to decrease water consumption as much as possible. CFD analyses for all models have been completed. The results have been examined comparatively, and they are presented as shown in the following Table 3.

Table 3. CFD results of diverter cap models

Model Name	Baseline	Model A	Model B	Model C	Model D	Model E
Total Pressure Loss Upper (mbar)	52	55	39	41	38	44
Total Pressure Loss Upper (mbar)	143	167	145	146	151	146
Water Volume (ml)	154.9	131.8	126	114.8	117.9	111.4

Within the scope of this study, various designs of the diverter cap used in models without diverters have been examined. The design currently used in production, known as Model A, as well as the baseline where no specific geometry is present, and new models aimed at reducing pressure losses and water consumption have had their pressure and velocity distributions analyzed.

Results and Discussion

After analyzing the results, the outcomes of Model A, currently in use, have been compared with those of the other models. The comparison data consist of pressure losses between the connection points of the spray arms and the entry to the diverter chamber, as well as the water volume within the diverter chamber. The difference in results for the other models compared to Model A is provided in the table below which is Table 4.

Table 4. CFD results of diverter cap models

Model Name	Model A	Model B	Model C	Model D	Model E
Total Pressure Loss Upper (mbar)	55	-16	-14	-17	-11
Total Pressure Loss Upper (mbar)	167	-22	-21	-16	-21
Water Volume (ml)	131.8	-5.8	-17	-13.9	-20.4

The analyses have been reviewed and considering both the reduction in pressure losses and the decrease in chamber volume, it has been decided to proceed with Model 2C. Firstly, the prototyping process has been completed using a 3D printer. Subsequently, laboratory tests have been conducted. The reduction of pressure losses has allowed for a higher flow rate at the same pump speed.

After completing the laboratory tests, the design of the component was finalized. The mold output of the component is shown in Figure 13 below.

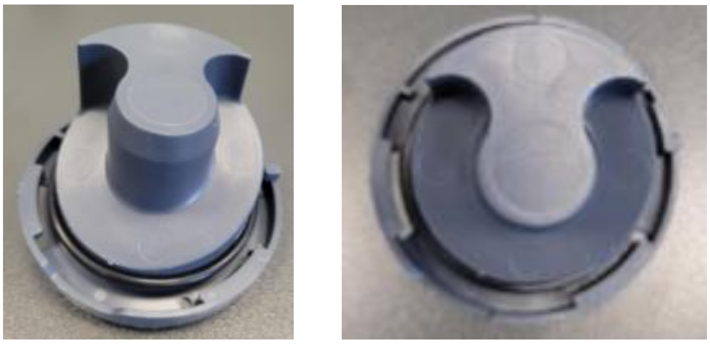

Figure 13. Output of the plastic injection mold after CFD study

Conclusion

With this study, a design change has been made for the diverter chamber. The design currently used in existing productions and other studies have been analyzed with CFD software at the same flow rates, and their results have been compared. The results have been examined to reduce pressure losses and minimize the empty volume of the chamber.

According to the analysis results, a part has been produced with 3D printers for the selected Model C. In tests conducted with this part, the total flow rate has increased from 38.2 l/min to 41.3 l/min at a pump speed of 2400 rpm, and from 52 l/min to 57 l/min at a pump speed of 2800 rpm. As a result of the change in the geometry of the diverter cap alone, there has been an approximate 8% increase in flow rate at 2400 pump speed and a 9% increase at 2800 pump speed.

Recommendations

Hydraulic circuit losses in hydraulic systems cause significant inefficiencies in the system. Hydraulic systems for dishwashers operating at high pump speeds should therefore be designed with this perspective in mind. HAD studies are very effective software tools to improve the design of relevant geometries. In our work, sometimes we need to examine a specific part of the system, and sometimes the entire system, through HAD analyses.

Scientific Ethics Declaration

The authors declare that the scientific ethical and legal responsibility of this article published in EPSTEM journal belongs to the authors.

Acknowledgements or Notes

* This article was presented as an oral presentation at the International Conference on Technology, Engineering and Science (www.icontes.net) held in Antalya/Turkey on November 16-19, 2023.

* For their contributions to this study, we extend our thanks to Renta Electric Home Appliances Industry and Foreign Trade Ltd. and our affiliated company, Haier Europe.

References

Ansys. (2010). *Ansys fluent user guide.* Conansburg: Ansys Inc
Batchelor, G. K. (1967). *An introduction to fluid dynamics*. Cambridge University Press.

DIN EN 50242/DIN EN 60436:2018-06. (2018) - *Electric dishwashers for household use - Methods for measuring the performance (IEC 60436:2004, modified + A1:2009, modified + A2:2012, modified*. VDE standarts

Livens, W. H. (1924). *Improvements in apparatus for washing household crockery and the like. FR579765*. UK : Intellectual Property Office.

Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*,*32*(8), 1598-1605.

Munson, B. R., Rothmayer, A. P., Okiishi, T. H., & Huebsch, W. W. (2012). *Fundamentals of fluid mechanics*.(7th ed.). Wiley.

Pritchard, P. J., & Mitchell, J. W. (2016). *Fox and McDonald's introduction to fluid mechanics*. John Wiley & Sons.

Wilcox, D. C. (1998). *Turbulence modeling for CFD* (Vol. 2, pp. 103-217). La Canada, CA: DCW industries.

Author Information

Vasıf Can Yildiran
Haier Europe- Renta Elektrikli Ev Aletleri San. Tic. Ltd. Sti.
26110 Odunpazarı Eskisehir, Turkey
Contact e-mail:*vasifcan.yildiran@haier-europe.com*

Fazıl Erinc Yavuz
Haier Europe- Renta Elektrikli Ev Aletleri San. Tic. Ltd. Sti.
26110 Odunpazarı Eskisehir, Turkey

Sebastian George Colleoni
Haier Europe
Via Privata Eden Fumagalli 20861 Brugherio MB
Italy

To cite this article:

Yildiran, V.C., Yavuz, F.E., & Colleoni, S.G. (2023). Investigation of the flow performance of the diverter cover in the dishwasher. *The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 26,* 462-473.