JMES The International Journal of
Mechanical Engineering and Sciences
https://iptek.its.ac.id/index.php/jmes
e-ISSN 2580-7471

Effect of Diffusers Installation in Inlet Primary Air Coal Pulverizer on Airflow Characteristic and Wear Concentration using CFD Modeling

R. Panji Satrio Wening Galih, Bambang Arip Dwiyantoro*

Department of Mechanical Engineering, Institut Teknologi Sepuluh Nopember, Sukolilo Surabaya 60111, Indonesia

Received: 20 January 2022, Revised: 24 August 2022, Accepted: 5 September 2022

Abstract

This study investigates the effect of installing several diffuser models in the supercritical power plant pulverizer inlet ducting on airflow characteristics and wear concentration. The pulverizer internal check results show that one area of the wall was wearing abnormally or faster than usual. This condition affects the availability of the pulverizer. Previous research has produced a method to overcome this phenomenon, but the design model was still unsuitable for the actual operational condition. This study simulated air motion in a pulverizer with six different diffuser models. The two variations in the number of blades were two and three blades, combined with three angle variations, which were 30°, 45°, and 50°. Viscous k-omega SST was used in this CFD modeling to simulate airflow from the primary inlet to the area above the throat ring. The results show the contours of the velocity of the air and the velocity vector on the pulverizer. From all the variations in this study, the 45° angle model with three blades and the 50° angle model with three blades can help overcome the concentration of wear on the pulverizer wall.

Keywords: Pulverizer, CFD modelling, airflow characteristic, wear, diffuser

1. Introduction

Adipala Supercritical Power Plant is a supercritical power plant located in Central Java, Indonesia. This power plant has a capacity of 660 MW which uses coal as fuel to heat boilers to convert water into pressurized steam. In the boiler system, a pulverizer function is to refine coal before it enters the boiler. The pulverizer type in Adipala Supercritical Power Plant is ZGM123GII. It is a medium-speed grinding roller pulverizer with a 123 cm average radius of grinding track [1]. There are six pulverizer units installed, each operating at a 20% MCR (Maximum Capacity Rate) capacity, which means that when the unit is at full load, five pulverizers are in operating condition, and one is on standby [2]. However, there is a condition where the unit operates the entire mill (6 pulverizers). The situation is when the performance of one of the pulverizers is not optimal or when coal as the fuel has poor quality. Therefore, maintaining the pulverizer's reliability is very important.

Adipala Supercritical Power Plant uses a pulverizer which, in its current operation, is experiencing a wear phenomenon concentrated in a one-liner wall area, almost opposite the direction of the inlet duct, as shown in Figure 1. The results of the internal check during periodic maintenance found that an area of the wall was wearing abnormally or faster than usual. This condition affects the availability of the pulverizer and potentially poses a fire hazard to the surrounding environment due to the leakage of fine coal.

Currently, efforts are being made to extend the life of the pulverizer components experiencing wear concentration by upgrading the pulverizer wall material to become more wear-resistant. Sukendar et al. [3] stated that it was necessary to upgrade the resistance of pulverizer wall liner material to the velocity of fine coal particles in the pulverizer to reduce the leakage damage of the pulverizer wall due to the concentration of wear on the body mill. Veranika et al. [4] also upgraded the material to increase the lifetime of the pulverizer wall. The material hardness value of the wall liner was increased. The pulverizer wall liner plate was modified into several segments for easy maintenance. However, the results were not optimal because the concentration of wear at one point in the area still occurs, although, in its application, it could reduce the rate of wear in that area. In addition, it increased the cost of pulverizer maintenance, which required purchasing new materials upgraded from worn-out components.

Apart from material upgrades, Wark [5] mentioned that installing a deflector just above the throat ring could protect the pulverizer from wear and abrasion. This deflector was made of material with better specifications than the previous pulverizer constituent material and was

---

*Corresponding author. Email: bambangads@me.its.ac.id.

formed into several segments. The purpose of this deflector was to deflect the air exiting the throat ring away from the pulverizer wall. Installing a deflector required a significant investment cost, and the costs for purchasing upgraded consumables were not cheap. There was a concern that the deflector could interfere with the original air movement pattern that needed to be maintained in the operation of the pulverizer.

Several previous studies have stated that the cause of the wear concentration that occurred on the pulverizer wall was the non-uniformity of airflow. Bhambare et al. [6] mentioned that the problem of operational disruption and the wear rate on the pulverizer was derived from the uneven airflow distribution in the throat area. Although the focus of the research was different, the results of the CFD simulation found that the primary air that flows into the pulverizer, after leaving the throat area, produced a zone of high velocity, which was opposite to the direction of the inlet duct.

Vuthaluru et al. [7], in 2005, stated that an even airflow distribution played an essential role in the coal transport process in the pulverizer. If the air distribution in the pulverizer was uneven, it caused several problems, one of which is excess air or the concentration of the amount of airflow at one point in the mill. The concentration of this airflow certainly caused large coal to be transported in the area, causing faster wear. Therefore, to see and study the condition of airflow distribution in the pulverizer, Vuthaluru et al. carried out a numerical simulation which can later be used as a basis for optimizing the performance of the pulverizer.

A year later, Vuthaluru et al. [8] conducted pulverizer modeling to determine the wear pattern due to variations in primary air velocity. As a result, the area with the highest velocity was in the throat area opposite the air inlet duct. This study used the Eulerian-Lagrangian CFD model to improve these conditions. The modification was to add baffle splitters to the inlet duct geometry of the pulverizer. It was known that increasing the total airflow on the left side of the baffle splitters and reducing the right side resulted in a more even distribution of air in the throat area. This condition certainly had the potential to prevent the concentration of wear on the pulverizer components [9].

From the literature above, adding a baffle splitter to the pulverizer's inlet duct could correct the flow's non-uniformity. Unfortunately, this modification could not be applied. In actual conditions, increasing airflow to the left side of the duct was impossible with only the baffle splitters model on the duct. The geometry of the duct needed to be changed too. However, changing the geometry was not the right choice because it cost a lot of money and took a long time to fabricate and install. So it was necessary to change the airflow direction without changing the geometry.

In addition to changing the airflow direction without changing the geometry, Huang et al. [10] added several blades to the intake channel of a ship's waterjet propulsion system. The blade was flat and arranged at a certain angle. The goal was to deflect the flow to the intake to increase the flow's uniformity. This method could effectively control the flow separation phenomenon and significantly affect hydraulic repairs without modifying equipment geometry. So, without changing the geometry, adding a few blades as diffusers with a certain number and angle could change the flow direction.

When discussing the design of the blade tilt angle, Awwad et al. [11] investigated the flow characteristics in a room through five-blade diffuser louver face ceilings using CFD. Several diffuser models were installed on 3D room models with variations in blade angles (60°, 65°, and 45°) and lip angles (0°, 5°, 10°, and 15°). A model with a 60° blade angle and 0° lip angle was installed on a 3D model with a variation of inlet return air position. Some of these variations were intended to see the distribution and characteristics of the flow produced in the room.

Based on several studies mentioned above, the researchers conducted a numerical study on the effect of inlet duct modification by adding a diffuser that had variations in the angle and number of blades on the characteristics of airflow distribution and the wear concentration on the pulverizer.

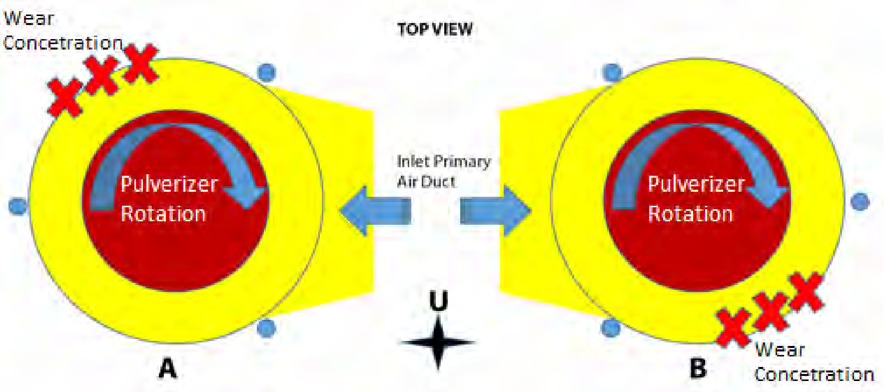


Figure 1. Wear concentration area on the pulverizer wall.

2. Method

This research started from the literature study stage by reviewing national and international journals, manual books, or other sources related to the research topic. This stage was crucial to help understand the problems raised in the research. The next step was to collect technical and operation data, then continued with geometry modeling, meshing, and initial simulation. After that, it was necessary to validate the model. Next, the simulation was designed. Lastly, the results obtained were analyzed and discussed. The flowchart of this research could be seen in Figure 2.

2.1. Geometry creation modeling and boundary condition

A pulverizer was one of the complex equipment in steam power plants. Detailed modeling took up a lot of time, both during the meshing process and during the solution iteration process. Therefore, in this study, a simplification of the geometric model was carried out. Bhambare et al. [6] mentioned that the uneven primary air condition in the mill in the throat ring area caused various problems in the pulverizer. Such as the excess air composition at one point of the pulverizer wall, which could cause damage. The location of the point of leakage or damage at the pulverizer wall was less than 2 meters from the location of the throat ring. Based on these conditions, the geometry was limited to starting only from the inlet ducting pulverizer to the area 2 meters above the throat ring.

Due to this simplification, the pulverizer's details, such as rollers, frames, and others, were not included in this modeling. The parts of the ZGM-type pulverizer could be seen in Figure 3, taken from the manufacturer's book [12].

Figure 2. Research flowchart.

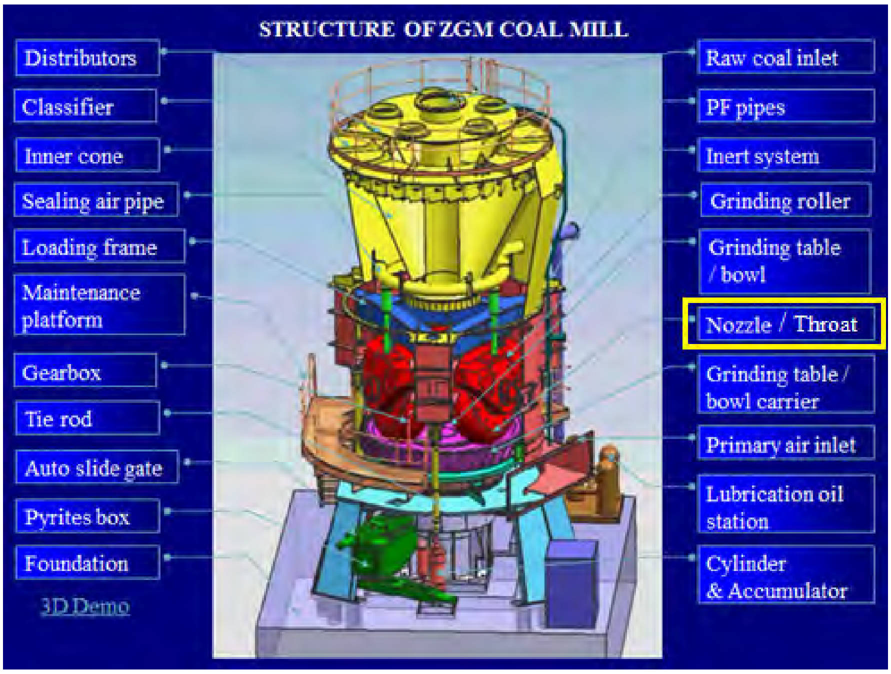


Figure 3. Structure of ZGM Coal Pulverizer.

Geometry models were based on the size of the existing company's technical drawings [13], shown in Figure 4 and Figure 5, using CAD software. The throat ring, which played an important role in this research, was made as detailed as possible. Detailed throat ring design could appropriately validate the simulation results. The blade on the throat ring had a slope of 33 degrees. The number of blades on the nozzle ring is 41 pieces. The three-dimensional domain model used the bottom-up approach because of the complexity of the pulverizer design. Figure 6 and Figure 7 showed an isometric view of the geometry and computational domain of this CFD simulation.

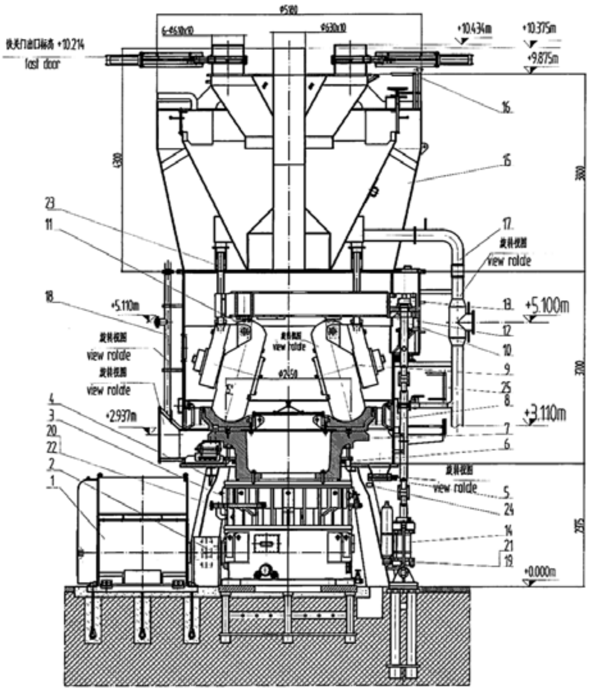


Figure 4. Adipala supercritical power plant ZGM pulverizer geometry.

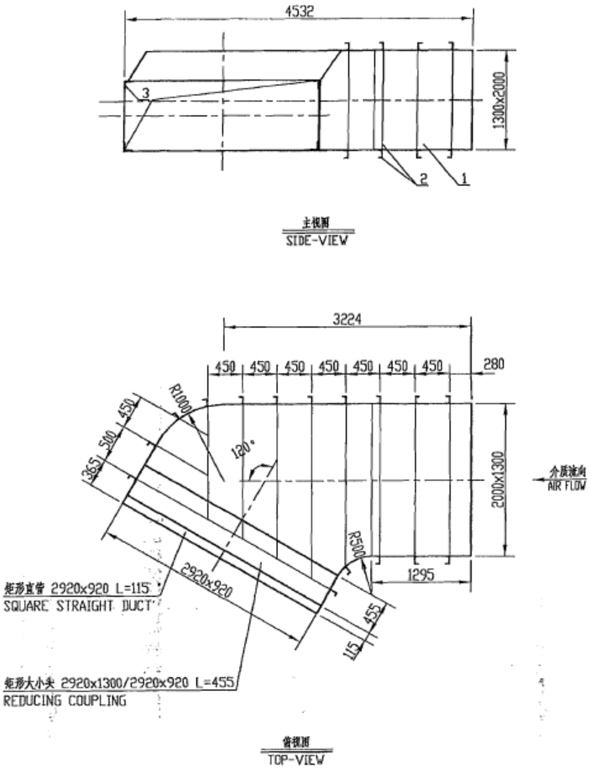


Figure 5. ZGM pulverizer primary air duct geometry.

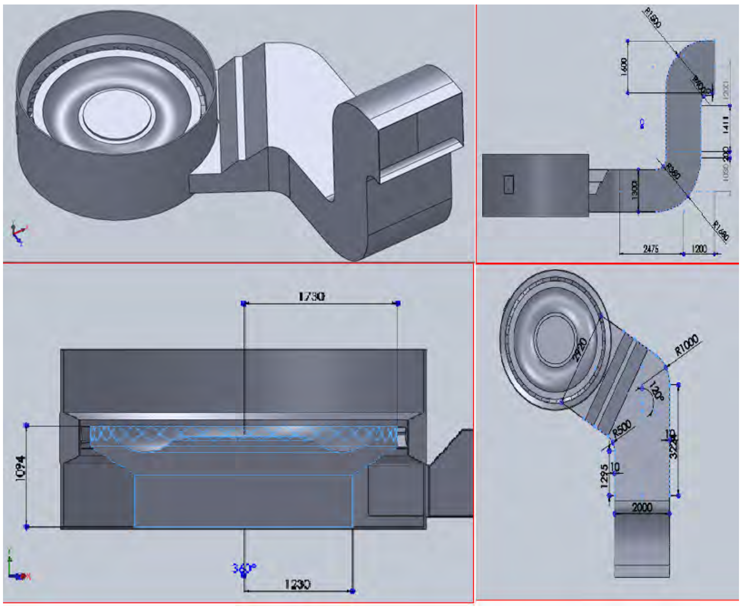


Figure 6. CAD drawing of pulverizer geometry.

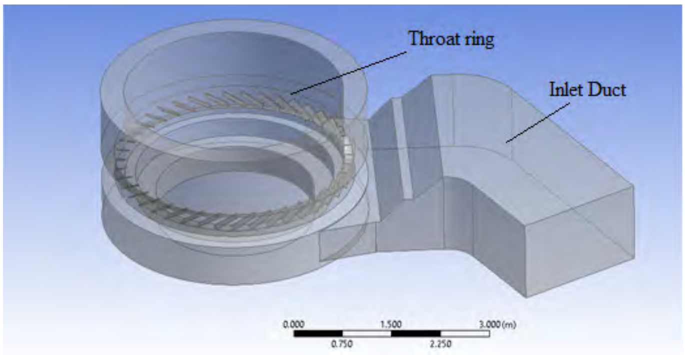


Figure 7. Computational domain of pulverizer.

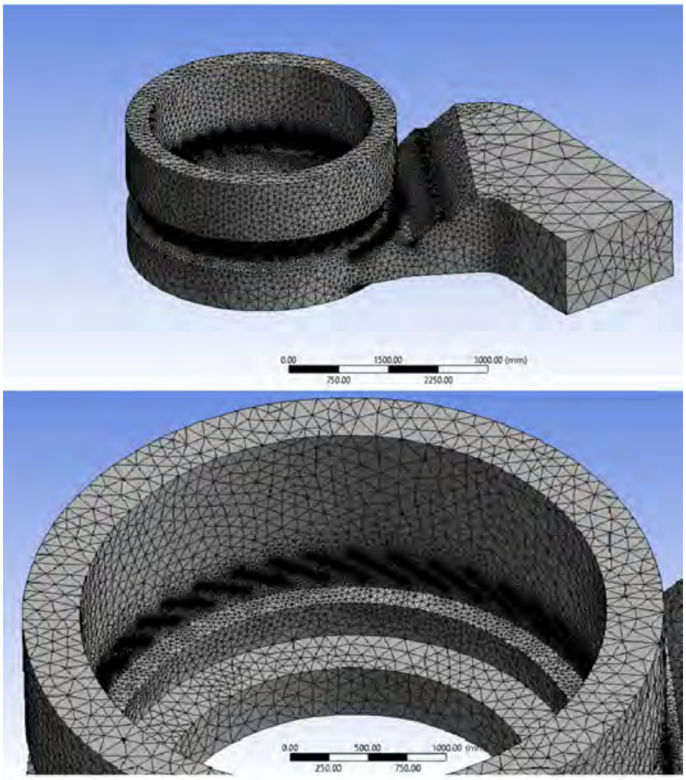


Figure 8. Meshing of computational domain.

ANSYS – a FLUENT pre-processor, was used for meshing. The three-dimensional computational domain was constructed from a collection of tetrahedral cells, as shown in Figure 8. The numerical model consisted of 754772 nodes and 3.768.990 elements. Mass flow and outflow boundary conditions were applied to the inlet and outlet. There was a moving rotation wall on this model with a clockwise (CW) direction. The rotation value was 28.8 r/min or 3.015929 rad/s [12] based on the technical data of the pulverizer. The primary airflow rate was 140 t/h or 38.888 kg/s, and the operational pressure was 6910 Pa.

The k-omega SST (Shear Stress Transport) model was used in this study with default setting conditions. This model had good behavior in adverse pressure gradients and separating flow. The SST k-ω model produced too large turbulence levels in regions with large normal strain, like stagnation regions and regions with strong acceleration. However, this tendency was much less pronounced than in a standard k-omega model [14]. For the numerical solver, pressured based solver was used. Velocity formulation was set as absolute, and gravitational acceleration was set in the Y-axis direction with a value of 9.81 m/s^2. Then, for the couplings between velocity and pressure, Semi-Implicit Method for Pressure-Linked Equation (SIMPLE) [15] algorithm, along with a second-order upwind scheme, was used for pressure, momentum, and turbulent setting. Therefore, the flow was incompressible, and the flow field was in a steady state.

2.2. Design Modification

The inlet ducting modification was done by adding variations in the angle of inclination and variations in the number of diffuser blades. This modification used a baffle splitter design, where the airflow left side composition condition was greater than the right side, which could uniform airflow distribution in the throat ring area. Therefore, variations in the number of diffusers were designed with the blade position tending to be on the right side of the ducting section. The angle variation was to direct the flow to the left side of the ducting section.

There were six variation models of diffusers used in this study. The variations used are two variations in the number of blades, which were 2 blades and 3 blades, combined with three variations in angles, which were 30 degrees, 45 degrees, and 50 degrees. The design used for the blade was a simple standard model with a square shape, as shown in Figure 9. This simple design was intended to make the fabrication process easier when applied to actual conditions. All variation models could be seen in Figure 10.

3. Results and Discussion

3.1. Model Validation

Validation of the model for the coal pulverizer was done by comparing the initial simulation results with indirect observations in the field. Validation was done by comparing the actual flow pattern formed on the pulverizer wall with the velocity vector of the initial simulation results in the wall area close to the throat ring. This pattern was observed when the mill stopped during operational standby or maintenance conditions because the pulverizer was one of the most complicated pieces of equipment in a power plant which was difficult to make observations and measurements when operating conditions [8].

The flow pattern formed on the pulverizer wall can be seen in Figure 11(a). Based on the results of the literature study, the wear pattern or flow pattern that arose was mainly due to the high air velocity that occurred in the pulverizer. The vector shown in Figure 11(a) indicated the direction of the airflow leaving the throat ring. The airflow angle created against the pulverizer wall matched the angle of the blade throat ring.

The initial simulation results produced a flow pattern in line with the results of the observation in the field, as shown in Figure 11(b). The figure showed a match between the direction of the airflow slope or the pattern formed on the pulverizer wall and the blade slope, both simulation results and actual observations in the field.

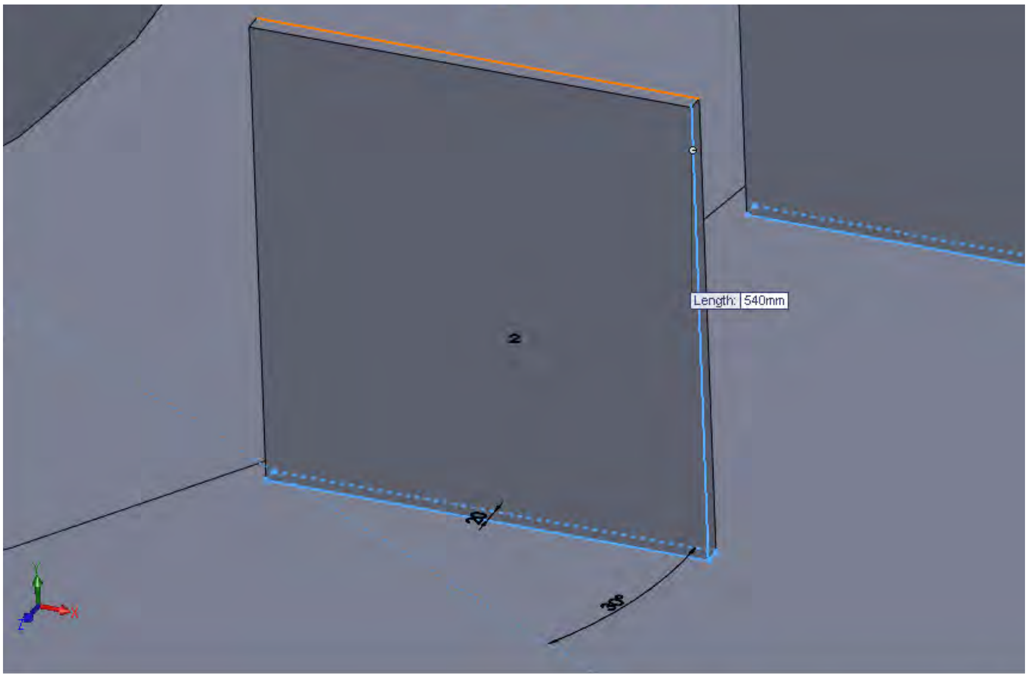
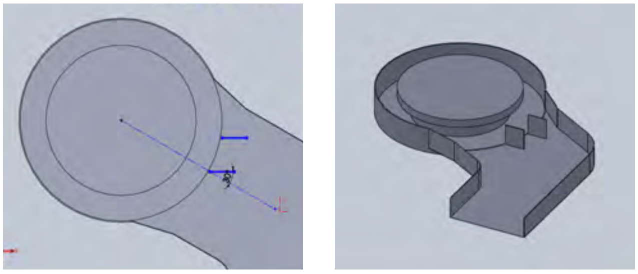
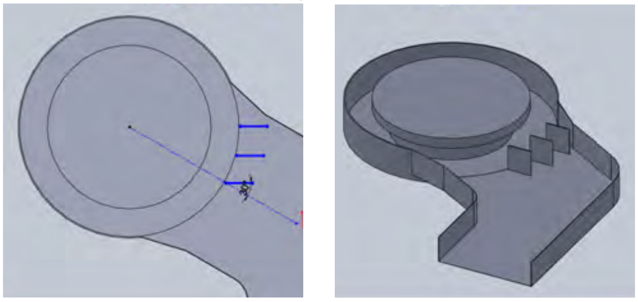


Figure 9. Diffuser blade shape and dimension.

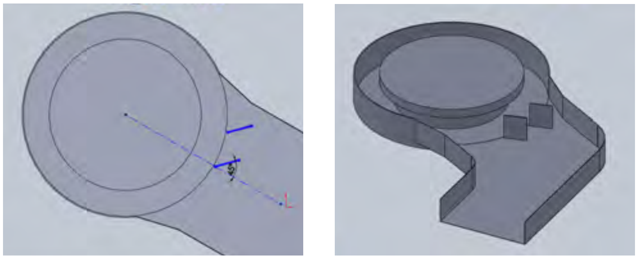
Model 1 - Angle 30°, 2 blades



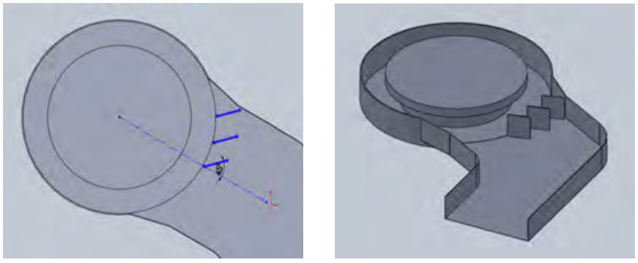
Model 2 - Angle 30°, 3 blades



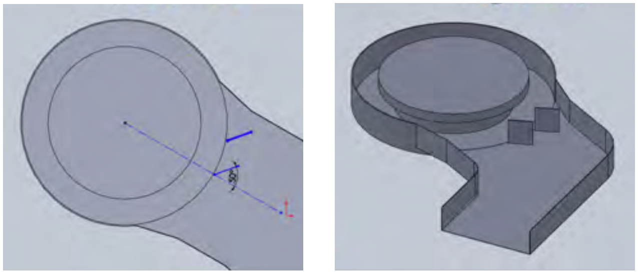
Model 3 - Angle 45°, 2 blades



Model 4 - Angle 45°, 3 blades



Model 5 - Angle 50°, 2 blades



Model 6 - Angle 50°, 3 blades

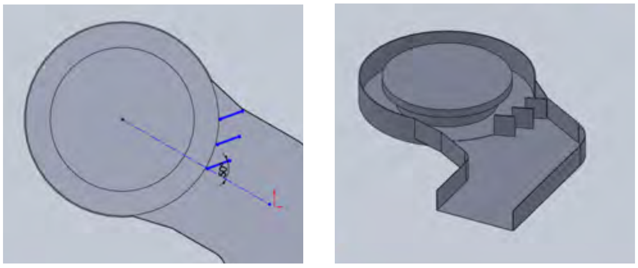


Figure 10. Six variations models of blade diffuser.

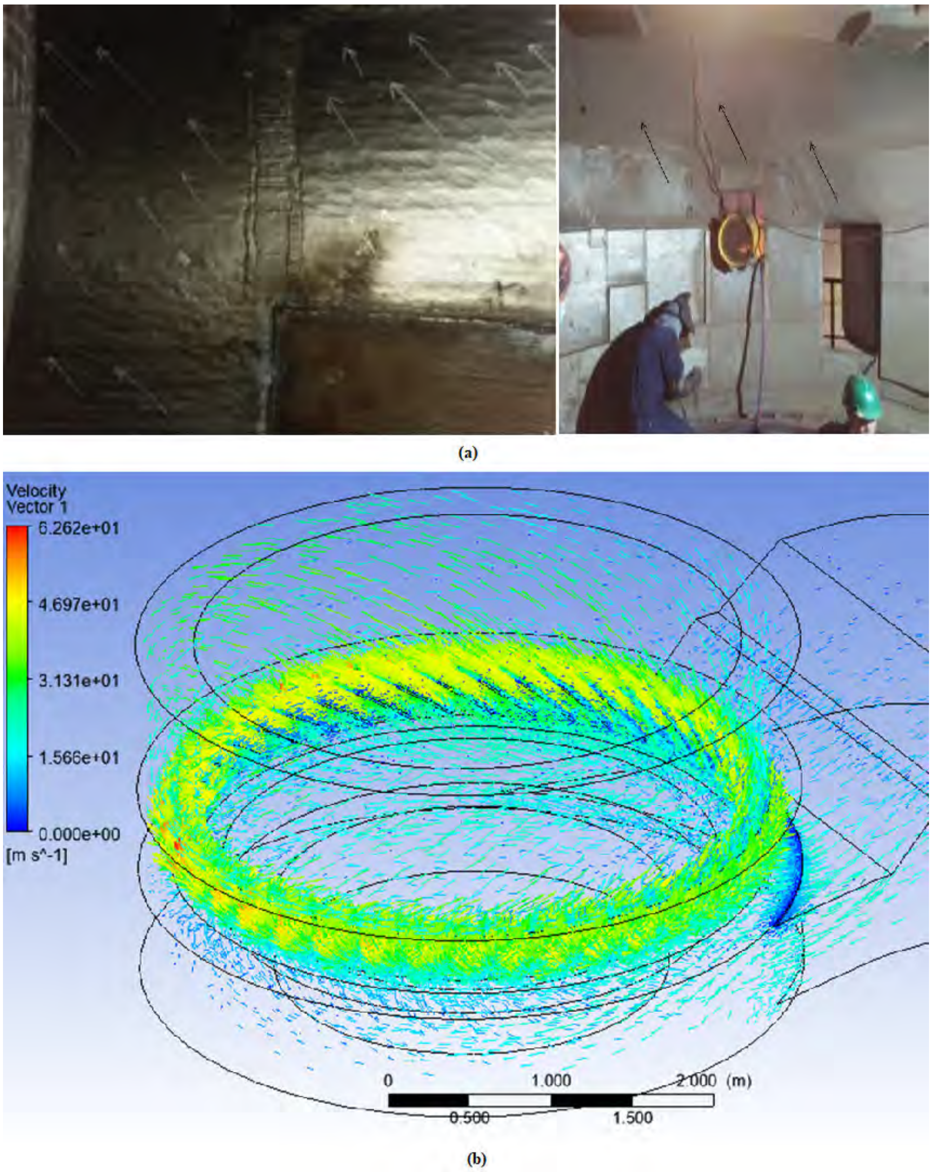


Figure 11. Comparison of actual observations with simulation results: (a)The results of the actual observation of the flow pattern or flow pattern formed by the flow of air coming out of the throat ring. (b) The results of the initial simulation vector velocity on the pulverizer.

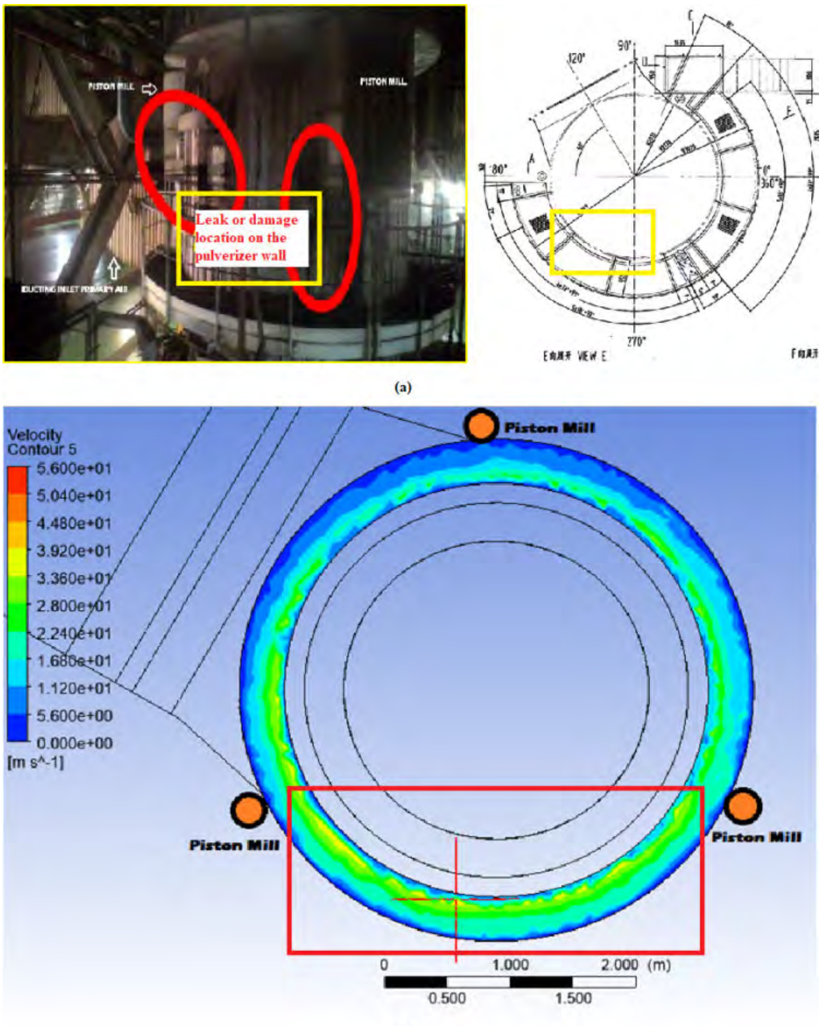
In addition to the suitability of the flow pattern between simulation results and actual observations in the field, the wear location between actual events and initial simulation was also checked, as shown in Figure 12.

Elevation $Y = 1.3$ m is the elevation on the pulverizer, which experienced the worst wear and tear based on the observations during the incident. Compared with the previously described condition in the field, the simulation results in Figure 12(b) showed the location of velocity concentration that was almost the same as the location of the actual leak in the field, which was almost opposite the inlet ducting. The velocity vector result in Figure 13 showed the same indication. Velocity concentration appeared in the area where the wall was leaking or damaged. With this condition, it could be said that the model made has been well validated.

In this preliminary simulation, post-processing of the velocity contour at an elevation of $y = 0.2$ m (the bottom of the pulverizer) and an elevation of $y = 0.955$ m (throat ring area) was also used as the basis for analysis for the simulation model.

3.2. Numerical Simulation Result

The simulation carried out in this thesis is to add a diffuser to the inlet ducting pulverizer. Six model variations have been made, where the changes made are from the angle and the number of blades applied. The following is a plot of the flow direction pattern and velocity distribution resulting from simulations carried out at elevations $Y = 0.2$ m, $Y = 0.955$ m, and $Y = 1.3$ m.



(a)

(b)

Figure 12. Areas of wear concentration (a) Actual description of conditions in the field (b) Initial simulation results of velocity contours which show that there was an area with velocity concentration at an elevation of $Y = 1.3$ m pulverizer.

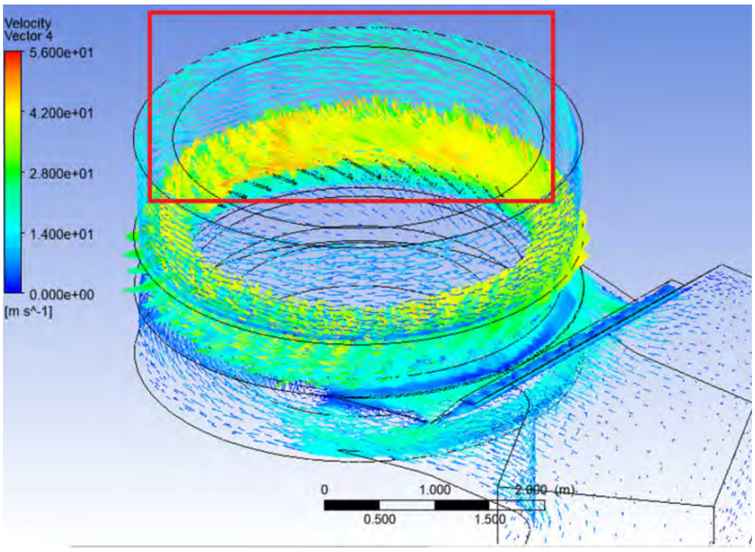


Figure 13. Velocity vector preliminary model.

3.2.1. Flow direction pattern under mill at elevation $Y = 0.2$ m

The following is a visualization of the velocity contour and vector to compare the flow distribution resulting from the preliminary and modification variation models. This comparison is intended to determine whether the flow is in accordance with the literature recommendation, which is to dominate the airflow to the left side of the inlet ducting to produce a uniform flow distribution in the throat ring area.

Before the diffuser model is added, the airflow path from the ducting inlet flows towards both sides (left and right) of the inlet. The air moves in a straight line following the cross-sectional trajectory in its path. Figures 15 and 16 of the preliminary model show how the airflows move toward the bottom of the pulverizer. After meeting the pulverizer shaft, the flow is divided into 2, to the left and right of the bottom pulverizer. A diffuser model is applied to direct the flow to the left side of the bottom of the mill. The angle variation changes the airflow trajectory to the desired direction. The simulation vector and contour results of various models in Figures 15 and 16, for modifications with an angle of 30 degrees, both with a combination of 2 blades and 3 blades, the flow direction still flows to the left and right sides of the bottom area of the mill. These dimensions and angles applied are unable to change the airflow trajectory. For the modification with an angle of 45 degrees, in the combination of 2 blades, the flow direction has indeed flowed more to the left side, but the flow to the right side is still there, so it is not said to be optimum. Meanwhile, in the combination of 3 blades, the flow direction is sufficient to meet the desired criteria, i.e., dominant flow to the left side of the area under the pulverizer. In the model with an angle of 50 degrees, the results are relatively similar to the model angle of 45 degrees, where the combination of 3 blades has been able to meet the criteria for the desired flow pattern. Based on the simulation results and visually analysis, it is concluded that the use of a blade angle of 45 and 50 degrees are sufficient to change the flow direction pattern at the bottom of the pulverizer ($Y = 0.2$ m) to be dominant towards the left side of the inlet ducting, with the choice of the number of blades as much as three better based on the formation according to the modified design.

3.2.2. Velocity distribution at elevation $Y = 0.955$ m

This elevation is the position in the upper throat ring pulverizer. Based on the literature study results, the non-uniform airflow velocity distribution in the pulverizer at this location is one of the references for good fluid flow performance in the pulverizer. Therefore, at this elevation, the value of the resulting velocity distribution is taken to determine the resulting flow characteristic pattern.

The velocity distribution value data is taken by numbering the space formed by the blade arrangement on the pulverizer so that 41 spaces are formed. In each of these spaces, three auxiliary lines are made. The line in this method is a tool in the post-processing step in ANSYS fluent in helping know the value at the desired spot. The first line (line 1 or *L1*) is the line close to the pulverizer shaft, and the second line (line 2 or *L2*) is the line in the middle, between line 1 and line 3. The last one, the third line (line 3 or *L3*), is the line that is closer to the outer diameter of the pulverizer throat ring. From these three lines, the average value of the velocity flowing in the space between the blades is taken (Figure 14).

The velocity value data is taken based on the velocity contours generated from the simulation, as can be seen in Figure 17. The velocity distribution data for each contour is obtained using the auxiliary line method. The data is then presented in the form of a graph.

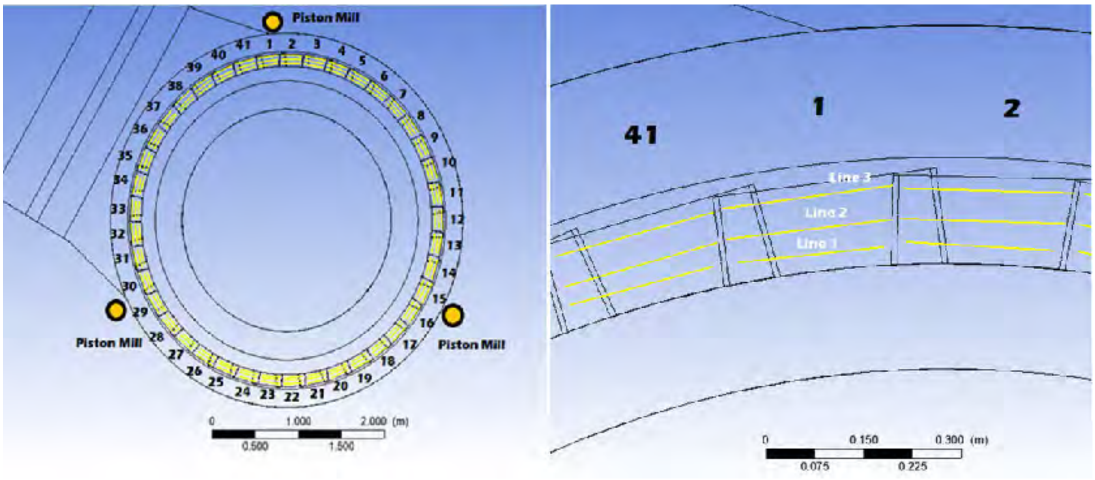
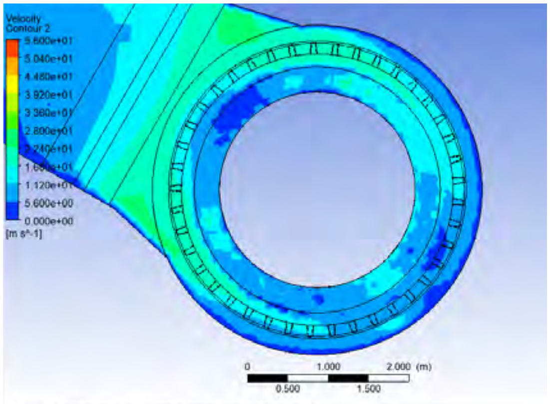
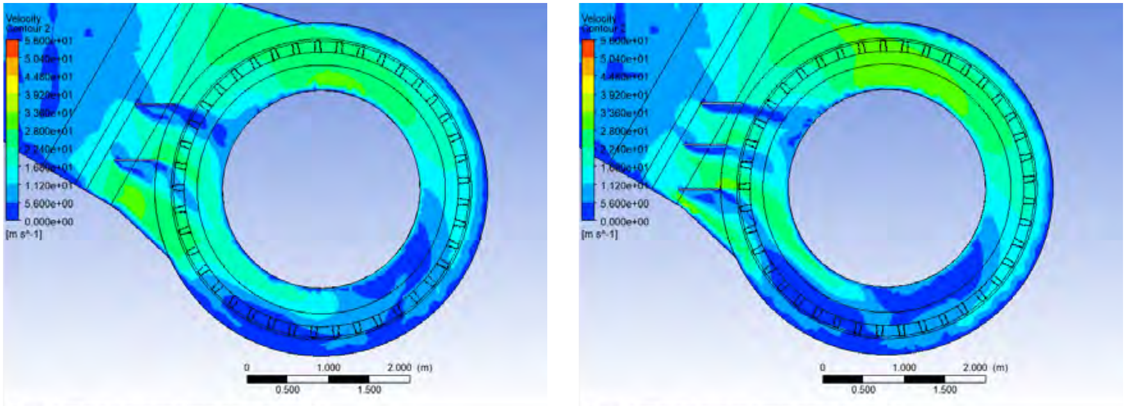
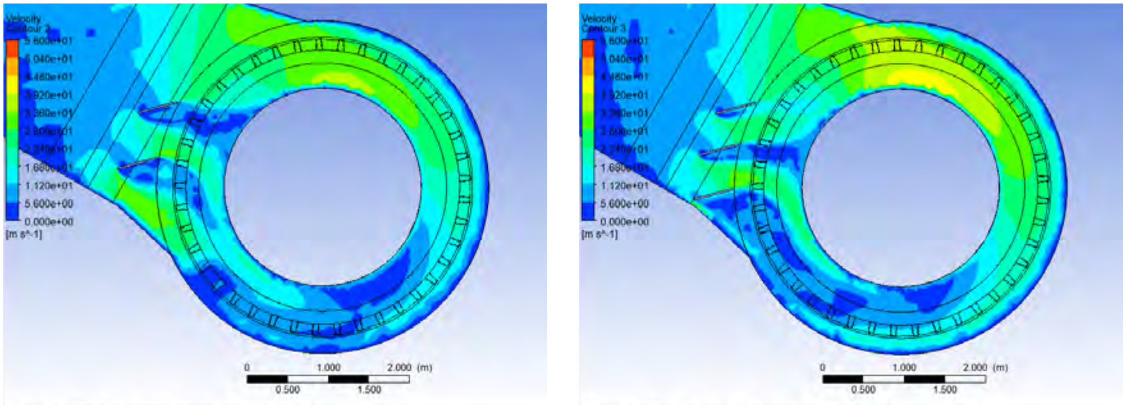


Figure 14. The blade space numbering mechanism and auxiliary line formation for speed data retrieval.


Preliminary Model


Model 1, Angle 30°, 2 Blades **Model 2, Angle 30°, 3 Blades**


Model 3, Angle 45°, 2 Blades **Model 4, Angle 45°, 3 Blades**

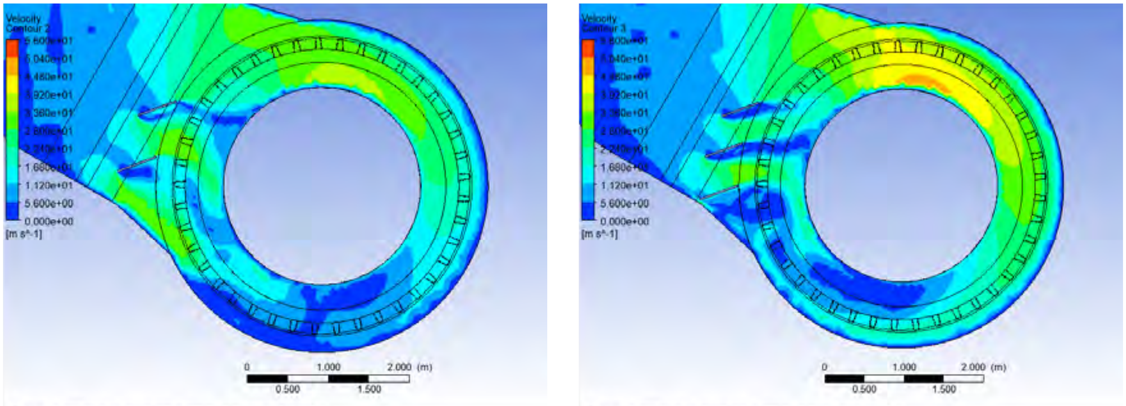
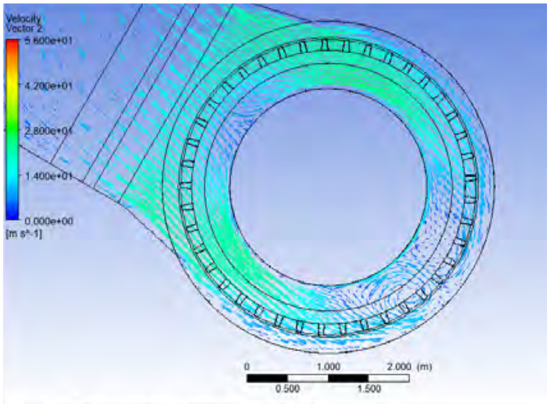
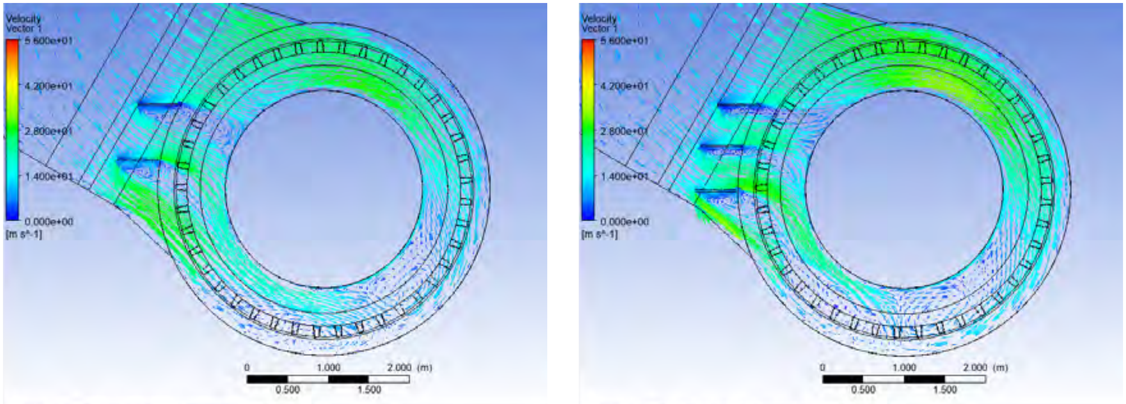

Model 5, Angle 50°, 2 Blades **Model 6, Angle 50°, 3 Blades**

Figure 15. Velocity contour pattern of flow direction at elevation $Y = 0.2$ m.

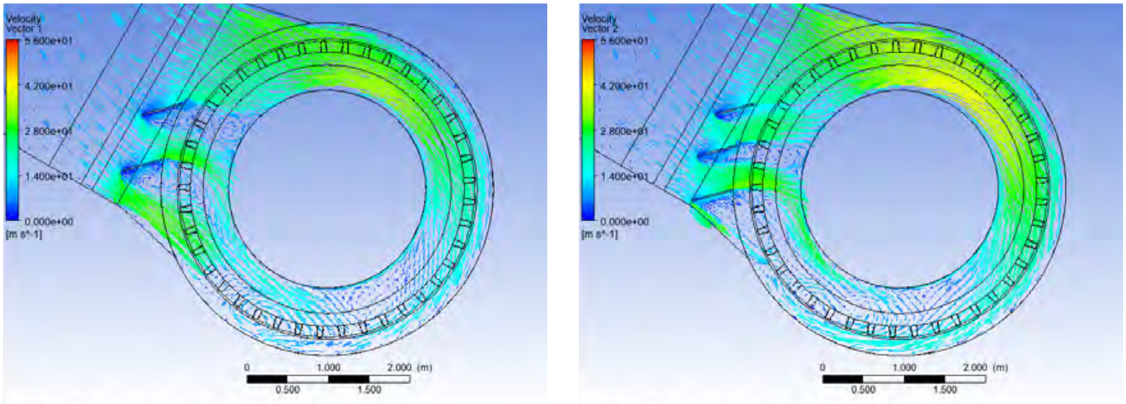


Preliminary Model



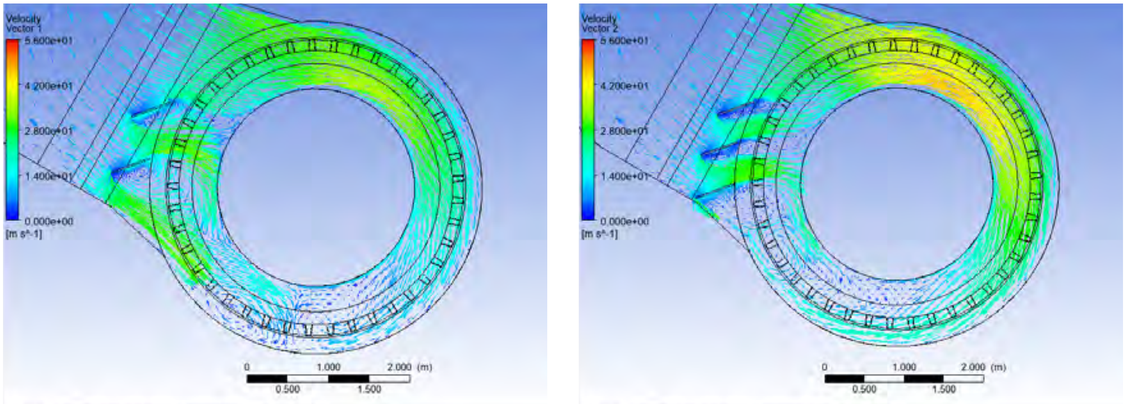
Model 1, Angle 30°, 2 Blades

Model 2, Angle 30°, 3 Blades



Model 3, Angle 45°, 2 Blades

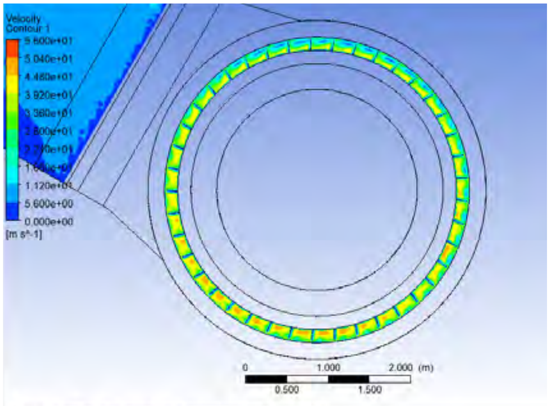
Model 4, Angle 45°, 3 Blades

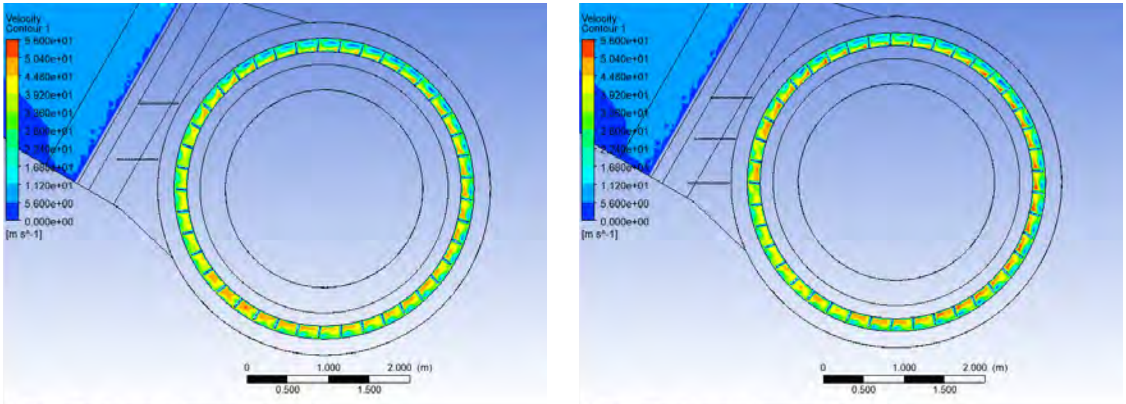


Model 5, Angle 50°, 2 Blades

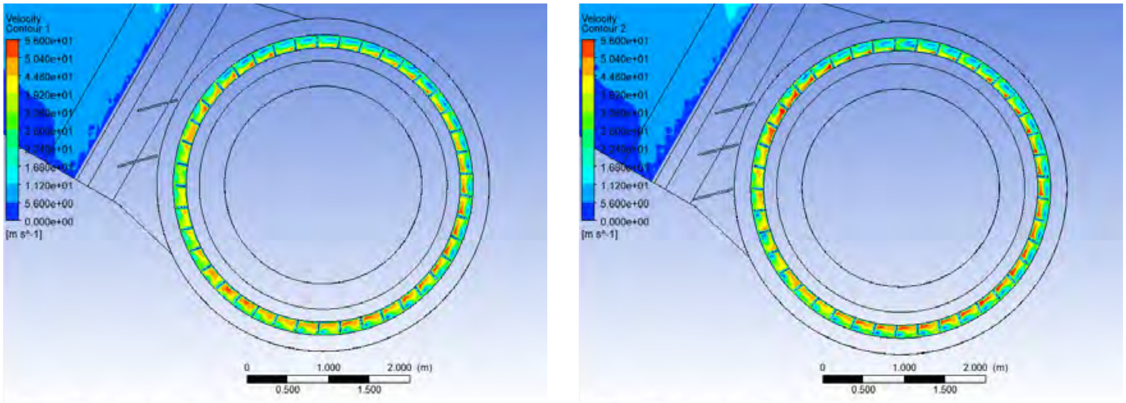
Model 6, Angle 50°, 3 Blades

Figure 16. Velocity vector pattern of flow direction at elevation $Y = 0.2$ m.

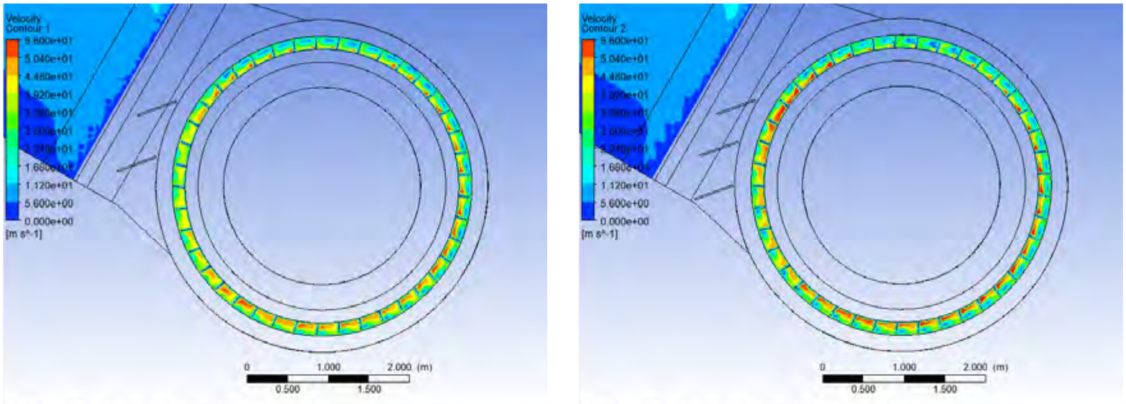

Preliminary Model


Model 1, Angle 30°, 2 Blades

Model 2, Angle 30°, 3 Blades


Model 3, Angle 45°, 2 Blades

Model 4, Angle 45°, 3 Blades


Model 5, Angle 50°, 2 Blades

Model 6, Angle 50°, 3 Blades

Figure 17. The velocity distribution contour at $Y = 0.955$ m elevation.

In Figure 18, the graph of the preliminary model shows that each auxiliary line has a different velocity level. Line 1, closer to the pulverizer shaft, has an average velocity value ranging from 34 m/s to 45 m/s. The average velocity on line 1 for the initial model shows an even distribution. However, this is not followed by the average speed value in lines 2 and 3. For line 2, the average velocity range is 23 m/s to 44 m/s, and for line 3 the average velocity range is 10 m/s up to 39 m/s.

Based on the observations and field validation results, it is known that the areas experiencing wear concentration are numbers 17 to 28, following the numbering of the blades in Figure 14. From the graph in Figure 18, the area between numbers 17 to 28 is an area that has an overlapping relationship between line 1 and line 2. Then for line 3, the average speed difference with overlapping line 1 and line 2 is 2 m/s to 11 m/s. These two things can be used to analyze the wear concentration on the pulverizer wall. Both must be met, not just one.

The presence of excess air that collects at high speed in a certain area of the pulverizer can cause large coal particles to be carried away and erode nearby walls. This condition occurs in areas located at numbers 17 to 28 as shown in Figure 18. The sections on lines 1 and 2 coincide, as they have nearly the same air velocity value. Line 3 has an average speed difference range value, as previously mentioned. A large amount of air collects and flows through this area, or in other words, it is concentrated in that area, causing the area to have a faster wear rate because of the large number of coal particles that are still large in size carried by the airflow and erode nearby area.

After plotting the graph of the existing modeling variant simulation, several results are obtained where there are graphs that meet the requirements for the occurrence of wear based on the graph observations in Figure 18. The graphs that meet the requirements for the occurrence of wear concentration based on the results of the initial modeling simulation are the graphs for the simulation of model 1 (Figure 19), model 2 (Figure 20), model 3 (Figure 21), and model 5 (Figure 23). The graphics simulation results of models 4 and 6 (Figure 22 and Figure 24) do not meet the criteria obtained in the discussion of the preliminary modeling simulation results, so that these models can prevent wear concentrations. The overall results of the analysis can be seen in more detail in Table 1.

Table 1. Results of data analysis variation model 1 to model 6.

No.	Variation	Line 1 & Line 2 Overlapping	(Line 1-Line 3) & (Line 2-Line 3) difference is between 2 m/s - 11 m/s	Conclusions
1	Model 1, Angle 30° , 2 Blades	Yes	Yes	There is a concentration of wear in areas no.25-32 and no.34
2	Model 2, Angle 30° , 3 Blades	Yes	Yes	There is a concentration of wear in areas no.25-32 and no.34-35
3	Model 3, Angle 45° , 2 Blades	Yes	Yes	There is a concentration of wear in areas no. 27-28 and no.33
4	Model 4, Angle 45° , 3 Blades	No	Yes	There is no wear concentration
5	Model 5, Angle 50° , 2 Blades	Yes	Yes	There is a concentration of wear in areas no.27 and no.34-35
6	Model 6, Angle 50° , 3 Blades	Yes	No	There is no wear concentration

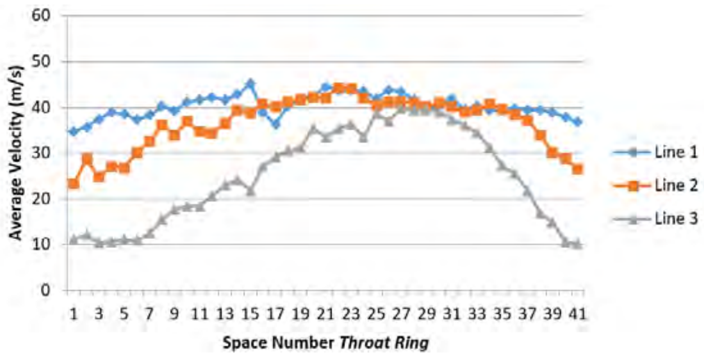


Figure 18. Velocity distribution graph at $Y = 0.955$ m, preliminary model simulation.

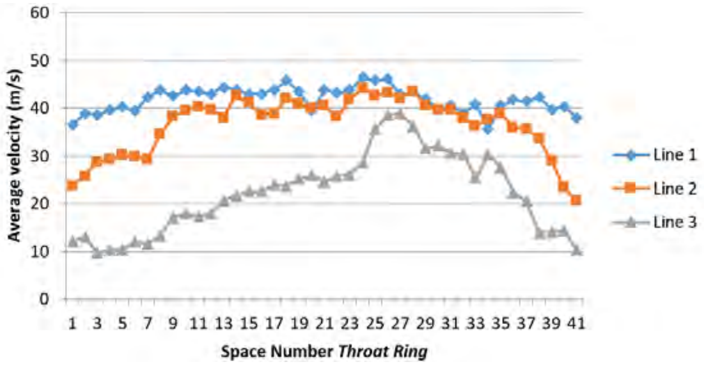


Figure 19. Velocity distribution graph at $Y = 0.955$ m, Model 1, angle 30°, 2 blades.

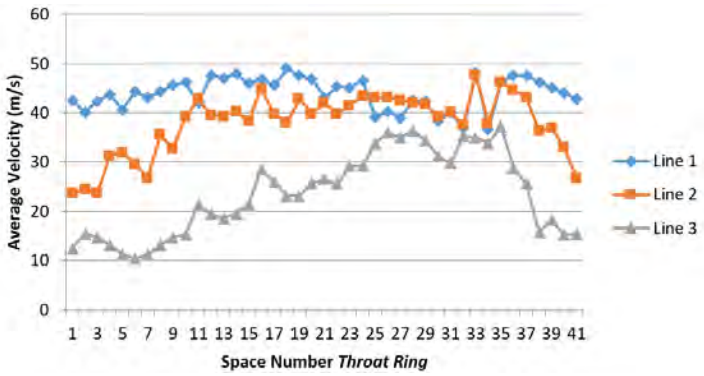


Figure 20. Velocity distribution graph at $Y = 0.955$ m, Model 2, angle 30°, 3 blades.

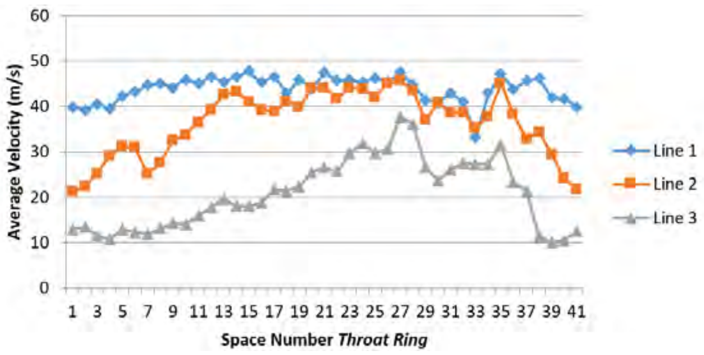


Figure 21. Velocity distribution graph at $Y = 0.955$ m, Model 3, angle 45°, 2 blades.

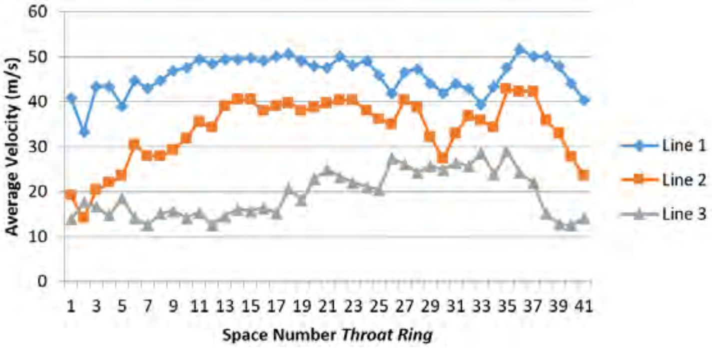


Figure 22. Velocity distribution graph at $Y = 0.955$ m, Model 4, angle 45°, 3 blades.

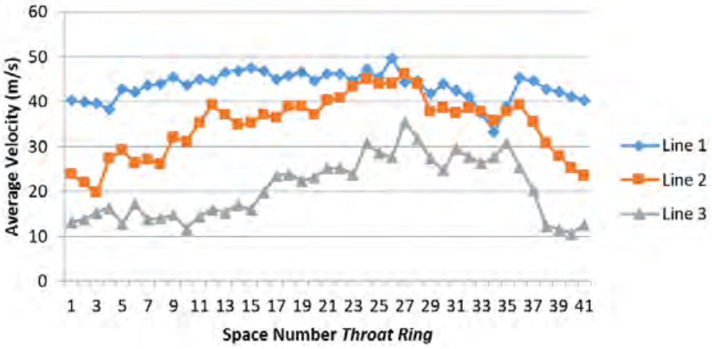


Figure 23. Velocity distribution graph at $Y = 0.955$ m, Model 5, angle 50°, 2 blades.

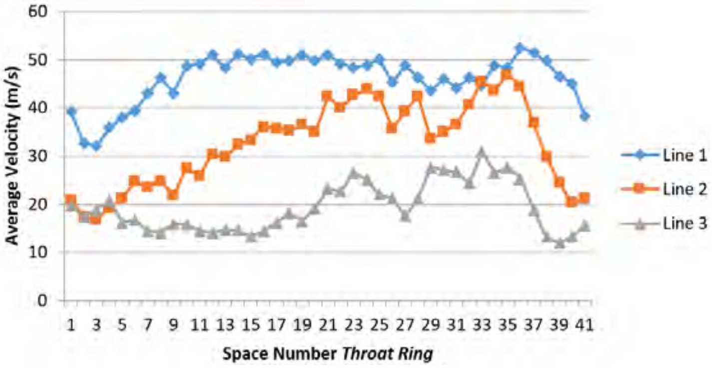


Figure 24. Velocity distribution graph at $Y = 0.955$ m, Model 6, angle 50°, 3 blades.

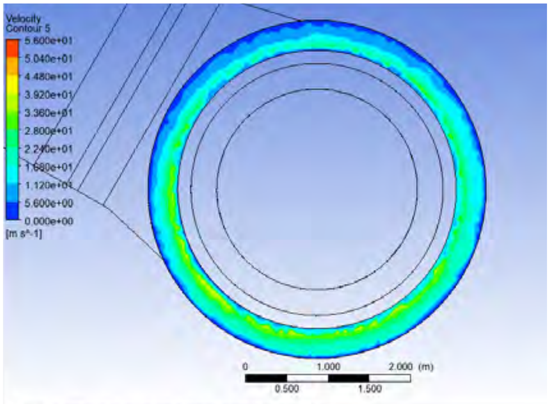
3.2.3. Velocity distribution at elevation $Y = 1.3$ m

Elevation $Y = 1.3$ m is where the wear concentration occurs on the pulverizer. Velocity contour depicts what occurs at this level apart from the elevation $y = 0.95$ m previously discussed.

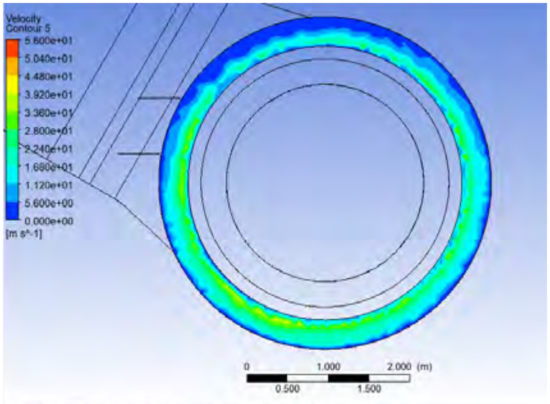
By increasing the elevation level to $Y = 1.3$ m, in the preliminary model shown in Figure 25, there is a contour with speed greater than 11.2 m/s touching the pulverizer wall where the wear concentration occurs. So we can use it as a reference to visually analyze the contours generated from the simulation models 1 to 6.

For the simulation results of model 1, we can see that the velocity contour with a value greater than 11.2 m/s is still touching the wall of the pulverizer. This indicates that there is still a concentration of wear on the pulverizer. This is also still found in the simulation results of model 2, model 3, and variation model 5. Simulation model 4 and model 6 showed different results, where the near-surface of the pulverizer wall was covered with velocity contours less than 11.2 m/s. This picture is in accordance with the results of the discussion of the value of the velocity distribution at an elevation of $y = 0.955$ m previously, where the simulation of variation model 4 and variation model 6 concluded that there was no concentration of wear.

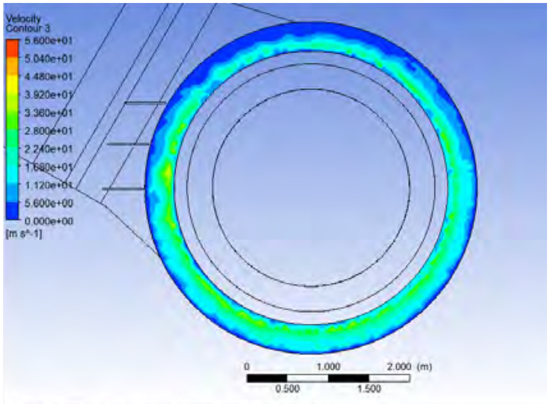
Preliminary Model



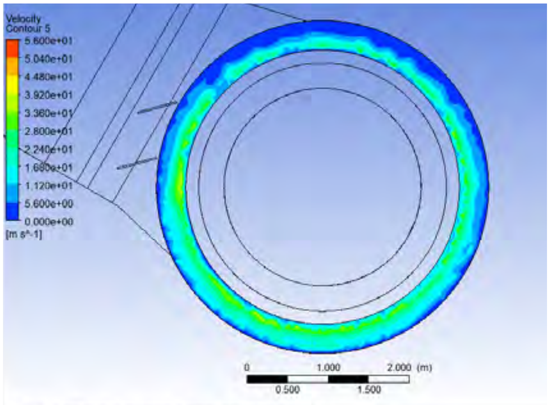
Model 1, Angle 30°, 2 Blades



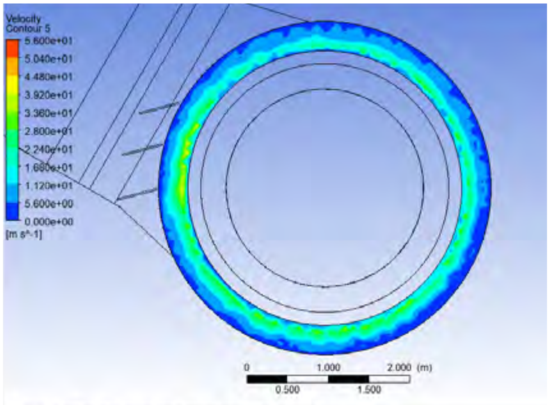
Model 2, Angle 30°, 3 Blades



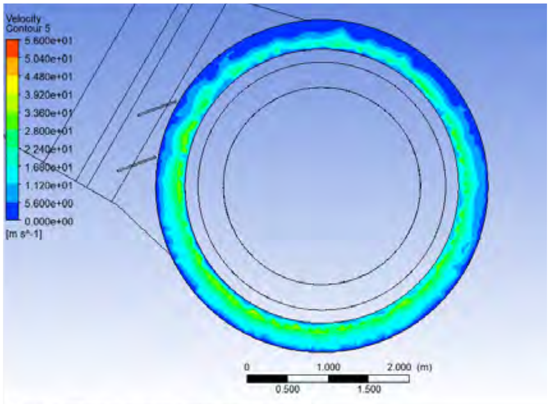
Model 3, Angle 45°, 2 Blades



Model 4, Angle 45°, 3 Blades



Model 5, Angle 50°, 2 Blades



Model 6, Angle 50°, 3 Blades

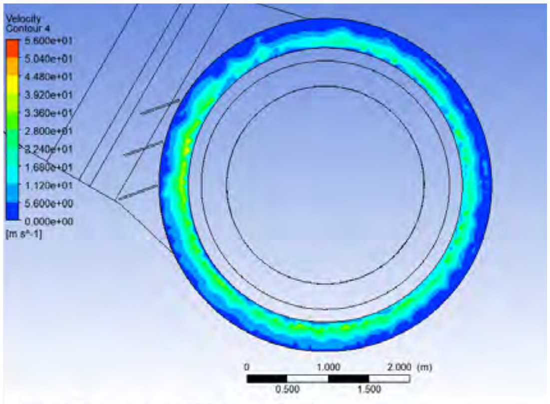


Figure 25. The velocity distribution contour at $Y = 1.3$ m elevation.

4. Conclusion

From the research and simulations that have been carried out, the flow direction pattern generated by the diffuser variation model with an angle of 45 and 50 degrees can change the flow at the base pulverizer to be dominant towards the left of the inlet ducting, with a choice of 3 blades which is better. Then on the results of the analysis of the velocity distribution, from all the diffuser variations model (six variations) that have been carried out in this study, model 4 (angle 45°, 3 blades) and model 6 (angle 50°, 3 blades) can help overcome the concentration of wear on the pulverizer wall.

References

[1] China National Technical Import & Export Corporation, “Operation and Maintenance Instructions for ZGM123 Roller Mill,” *Beijing Power Equipment Group*, 2012.

[2] China National Technical Import & Export Corporation, “Training manual for PLTU 2 Jateng 1x660 MW Adipala Cilacap (Boiler Part),” *Jiangxi Nanchang Power Plant*, 2013.

[3] I. Sukendar, A. Syahkroni, and B. T. Prasetyo, “Analisa kebocoran pada body mill/pulverizer menggunakan metoda root cause failure analysis (RCFA),” *Applied Industrial Engineering Journal*, vol. 3, no. 2, pp. 12–21, 2019.

[4] R. M. Veranika, M. A. Fauzie, and M. I. Badil, “Modifikasi plate liner pada air chamber pulverizer dengan bahan late wear resistance ARBEX 400,” *TURBULEN Jurnal Teknik Mesin*, vol. 1, no. 2, pp. 100–106, 2018.

[5] R. E. Wark, “Deflector for coal pulverizer/classifier,” Sept. 5 2006. US Patent 7,100,853.

[6] K. S. Bhambare, Z. Ma, and P. Lu, “CFD modeling of MPS coal mill with moisture evaporation,” *Fuel Processing Technology*, vol. 91, no. 5, pp. 566–571, 2010.

[7] H. Vuthaluru, V. Pareek, and R. Vuthaluru, “Multiphase flow simulation of a simplified coal pulveriser,” *Fuel Processing Technology*, vol. 86, no. 11, pp. 1195–1205, 2005.

[8] R. Vuthaluru, O. Kruger, M. Abhishek, V. Pareek, and H. Vuthaluru, “Investigation of wear pattern in a complex coal pulveriser using CFD modelling,” *Fuel processing technology*, vol. 87, no. 8, pp. 687–694, 2006.

[9] A. Kurniastuti, *Studi Numerik Pengaruh Temperatur Udara Primer Terhadap Potensi Terjadinya Self Combustion Di Coal Pulveriser Mill*. PhD thesis, Institut Teknologi Sepuluh Nopember, 2015.

[10] C.L. Huang, R. Dai, and Z.L. Wang, “Effects of upstream vortex generators on the intake duct performance for a waterjet propulsion system,” *Ocean Engineering*, vol. 239, pp. 109–838, 2021.

[11] A. Awwad, M. H. Mohamed, and M. Fatouh, “Optimal design of a louver face ceiling diffuser using CFD to improve occupant’s thermal comfort,” *Journal of Building Engineering*, vol. 11, pp. 134–157, 2017.

[12] Beijing Power Equipment Group, “The mid-speed ZGM series vertical mill,” *Beijing Power Equipment Group*, 2010.

[13] China National Technical Import & Export Corporation, “ZGM123G-II coal mill technical drawing,” *Beijing Power Equipment Group*, 2012.

[14] A. Wicaksono and B. A. Dwiyantoro, “Numerical study of the effect of excess air variation and biomass amount at co-firing process for tangential coal pulverized boiler,” in *Recent Advances in Mechanical Engineering*, pp. 265–272, Springer, 2023.

[15] H. K. Versteeg and W. Malalasekera, *An introduction to computational fluid dynamics: the finite volume method*. Pearson education, 1995.