Journal Homepage: - www.journalijar.com

INTERNATIONAL JOURNAL OF ADVANCED RESEARCH (IJAR)

ISSN NO. 2320-5407

Article DOI: 10.21474/IJAR01/17808
DOI URL: http://dx.doi.org/10.21474/IJAR01/17808

RESEARCH ARTICLE

OPTIMIZATION DESIGN OF INLET AND OUTLET FOR A TURBINE POWER GENERATION EQUIPMENT

Wenxin Yang and Qilin Shu
Shenyang Ligong University School of Mechanical Engineering, Shenyang, Liaoning 110159, China.

Manuscript Info

Manuscript History
Received: 05 September 2023
Final Accepted: 09 October 2023
Published: November 2023

Key words:-
Numerical Simulation, Turbine Generation, Structural Optimization, Torque Coefficient, Air Inlets and Outlets

Abstract

To enhance the operational efficiency of turbine generators, this study takes the air intake structure of the U.S. military turbine fuzes and a self-designed structure as research subjects. By employing the numerical simulation software Fluent, we compared the turbine's efficiency under various structural parameters. The results indicate that the U.S. military's intake structure is not suitable for long intake duct types; the turbine's efficiency peaks when the diameter at the end of the intake duct is approximately 1.5 times that of the turbine's radius. Future improvements in efficiency could be achieved by adjusting the dimensions and the matching relationship between the turbine, the intake duct, and the internal flow field.

Introduction:-

Aerodynamic turbine technology, as a highly regarded topic within the field of wind energy utilization, has seen wind become a clean and sustainable source of electricity with the continuous development of renewable energy, finding widespread application in power generation and battery charging. Against this backdrop, fuze technology has begun to emerge, with its performance and reliability largely depending on the method of generating electrical signals[1]. Traditional methods of generating electrical signals have several disadvantages, including high energy consumption, susceptibility to external interference, and limited storage time. In contrast, aerodynamic turbine technology offers numerous advantages. Firstly, it harnesses renewable wind resources, providing a clean and sustainable energy source for the environment. Moreover, aerodynamic turbine technology possesses a high resistance to interference. Traditional methods are easily affected by external disturbances, whereas aerodynamic turbine technology, leveraging the natural properties of wind, exhibits strong stability and interference resistance during the signal generation process. This ensures that fuze technology can operate reliably even in harsh environments. Additionally, the electrical signals generated by aerodynamic turbines are capable of long-term storage. Traditional methods cannot meet long-term storage requirements, but signals produced by aerodynamic turbine technology can be stored in appropriate devices for future use, ensuring the long-term reliability of fuze technology. For these reasons, how to utilize aerodynamic turbines to generate electrical signals has become a research area of great interest[2].

Aerodynamic turbine technology represents an innovative solution strategy, with the primary goal of efficiently harnessing wind energy to generate electrical signals[3]. Within this technological domain, enhancing the turbine's operational efficiency has become a central issue of widespread concern among researchers, relating not only to the efficiency of energy conversion but also directly impacting the prospects for sustainable development applications. Chen Zhimin[4] and others have used a computational axisymmetric design method to conduct an in-depth study of

Corresponding Author:- Wenxin Yang
Address**:-** Shenyang Ligong University School of Mechanical Engineering, Shenyang, Liaoning 110159, China.

the turbine's outlet section, finding that a bicubic curve structure significantly improves overall efficiency. Shen Dezhang[5] and others, by optimizing the shape of an external turbine, achieved the function of delaying the release of fuze safety. XU Shenggang[6] and others improved the operational efficiency of an embedded turbine by adjusting the shape of the projectile and reducing the size of the step top. Li Aiting[7] used a coupling method of blade profile and flow field, employing a CFD solver to design new blades, thus enhancing the impeller's operational efficiency. Liu Yong[8] studied an embedded side intake turbine motor; through establishing mathematical models and conducting numerical simulations, he verified the correctness of the approach and proposed optimization suggestions, successfully increasing the turbine's rotational speed by more than 8%. However, the impact of the shape of the turbine's inlets and outlets on its operational efficiency has not been extensively studied. Therefore, this paper combines the direct intake duct structure of the U.S. military with a newly designed structure, using numerical simulation calculations to explore how to further improve the turbine's operational efficiency.

Research Subject

This paper focuses on how to improve the working efficiency of turbines by finely adjusting the air inlets and outlets and revolutionizing the types of air ducts, utilizing numerical simulation technology to optimize the torque experienced by the turbines. Especially during the operation of turbines, due to the wide range of wind speed variations they encounter, selecting an inlet and outlet condition that ensures stable operation of the turbine across a broad range of inflow velocities is particularly crucial. As demonstrated in Figure 1 and Figure 2, this study employs a convergent-divergent air duct design. Compared to the traditional straight duct design, this structure is more conducive to maintaining a stable operational speed of the wind turbine across a larger range of inflow velocities. Additionally, the design of the convergent-divergent air duct helps to reduce turbulence within the turbine[9], further enhancing the overall efficiency and lifespan of the turbine[10]. Through this design, it is expected to provide strong support and reference for the further development of turbine technology.

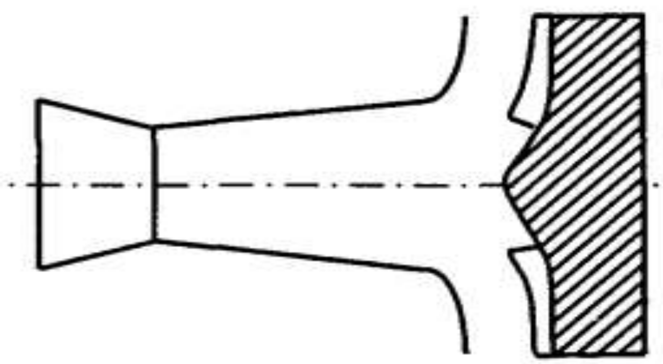

Figure 1:- Convergent-Divergent Type.

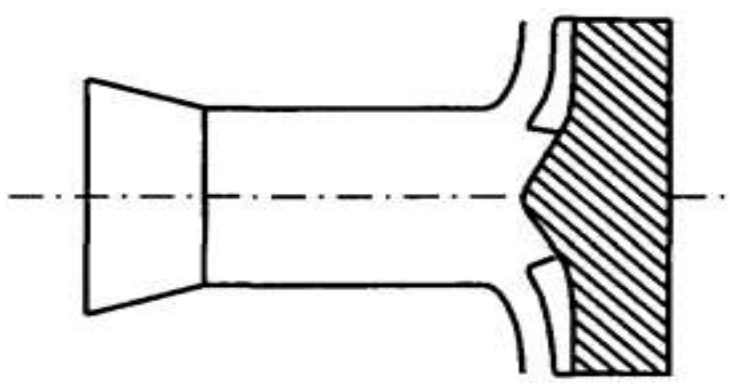

Figure 2:- Convergent Straight Tube Type.

Inlet and Outlet Numerical Simulation Analysis

Three-Dimensional Model and Finite Element Model

To conduct numerical simulation research, this paper utilized the Fluent three-dimensional numerical simulation software, specifically for a certain working condition. In the simulation calculations, to minimize the impact of the flow field domain size on the research subject, the inlet of the flow field domain was set at a distance of 20R from the turbine, the outlet at 50R, and the radial distance at 30R[11], as shown in Figure 3.

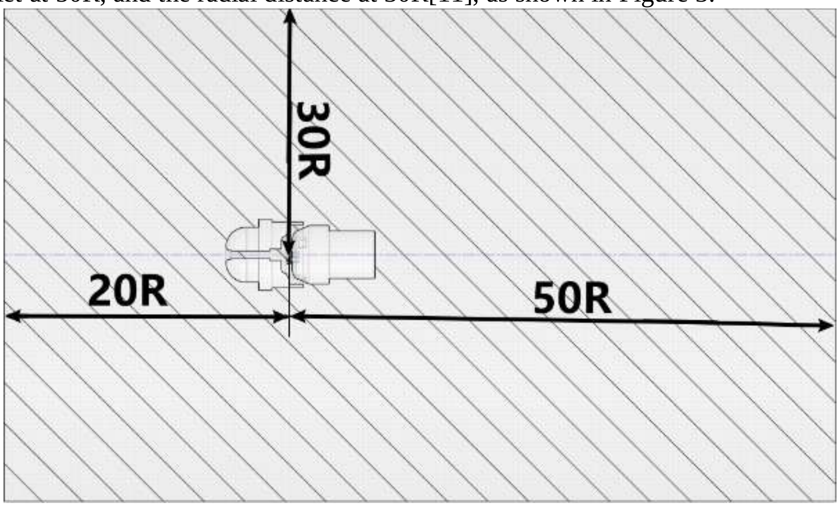

Figure 3:- Schematic Diagram of Flow Field Domain Size.

In this numerical simulation, the SST model was adopted. This is an improved k-ω turbulence model that combines the advantages of the k-ε and k-ω models. The model can accurately describe the flow characteristics near the wall and takes into account the transfer of shear stress, resulting in higher predictive accuracy in complex flow situations. Compared to other complex models, the SST model has a relatively lower computational cost[12]. In terms of simulation settings, based on the specific working condition, an incompressible, steady-state calculation method was used, with adiabatic, no-slip boundary conditions for solid walls[13]. The inlet and outlet temperatures and pressures were set values, the solver type was pressure-based, and the residual was set to 1e-4.

Grid Independence Verification and Grid Type

Considering grid independence, five different numbers of grids were tried: approximately 0.95million 2.12 million, 3.01 million, 4.77 million, and 5.97 million. By monitoring the axial force of the wind turbine, as shown in Figure 4, it was found that the axial force deviation of the last two grids was less than 1%. Therefore, based on a balance of computation time and accuracy, a grid refinement scheme with 4.77 million grid cells was selected.

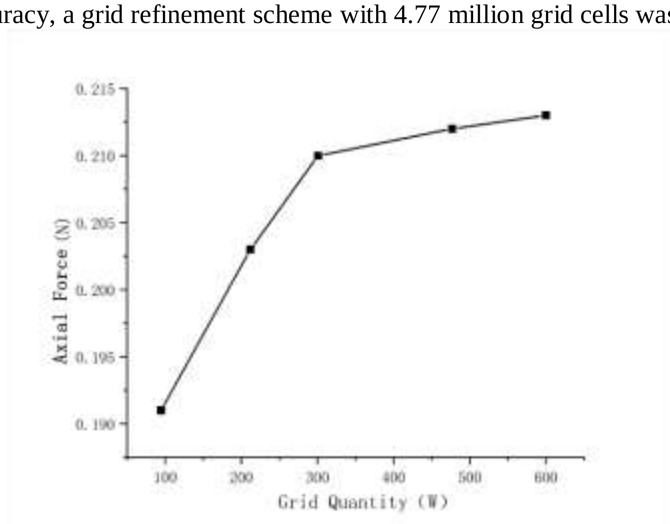

Figure 4:- Curve of Axial Force with Grid Quantity.

The calculation used a polyhedral grid type that coexists with structured and unstructured grids. Compared to tetrahedral and hexahedral grids, polyhedral grids have several advantages. First, they can accurately describe geometries with complex and sharp edges. Second, their grid quality is relatively better, without producing twisted, skewed, or elongated cells. This makes polyhedral grids more flexible and efficient when dealing with complex flow problems such as boundary layers and shock waves. Additionally, for the same size, polyhedral grids have fewer but higher quality cells. In Fluent software, the algorithms that support such grids are also relatively more efficient. Therefore, polyhedral grids were chosen as the grid type for this calculation. Specific local refinement and boundary layer settings can be seen in Figure 5 and Figure 6.

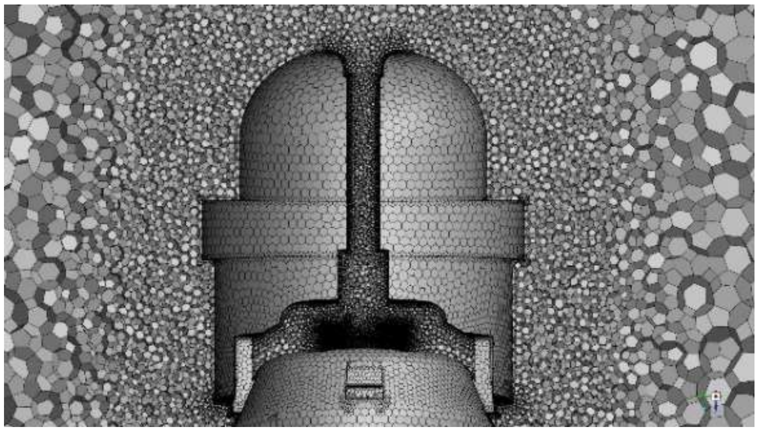

Figure 5:- Local Refinement Part of the Grid.

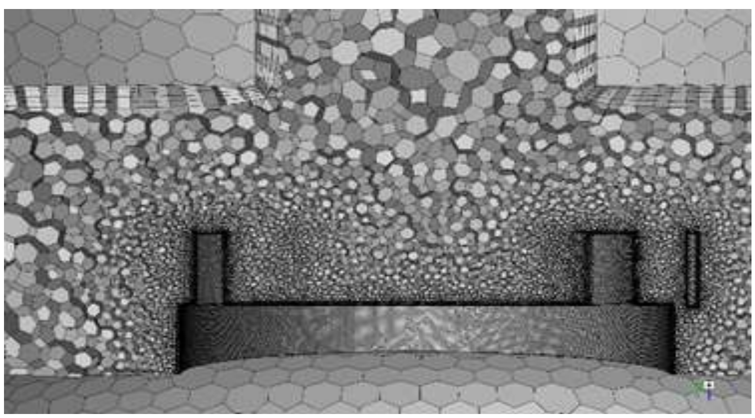

Figure 6:- Boundary Layer Part of the Grid.

Inlet and Outlet Optimization Design Scheme

Figure 7 presents a cross-sectional view of the U.S. military fuze turbine generator's direct intake duct[14]. Figure 8 further details various design schemes: part a is a schematic diagram of the original three-dimensional model and the air duct cross-section. Parts b and c are based on the U.S. military fuze's intake design, showing two different versions with and without a skirt at the outlet. As these types of air ducts are long intake duct types, a new design hypothesis is proposed. In this hypothesis, the diameter of the wind turbine remains unchanged, the design of the front half of the intake duct still refers to the U.S. military structure, but the latter half, especially the air duct diameter L near the wind turbine, will be adjusted according to the radius R of the wind turbine. The outlet is also divided into two types: with and without a skirt. Parts d and e of Figure 8 respectively show the skirtless and skirted schemes when L is 1.5R, while parts f and g are the corresponding designs when L equals R. Parts h and i further explore the skirtless schemes when L is 1.25R and 1.75R.

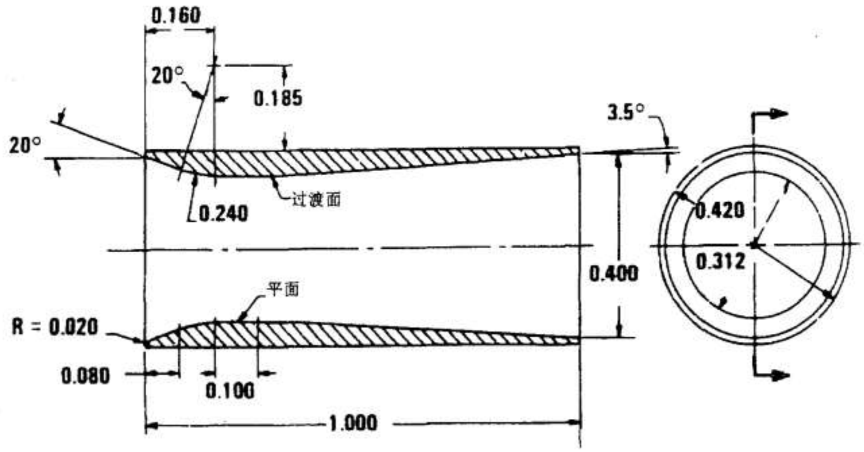

Figure 7:- Intake Duct Diagram of the U.S. Military Fuze Turbine Generator.

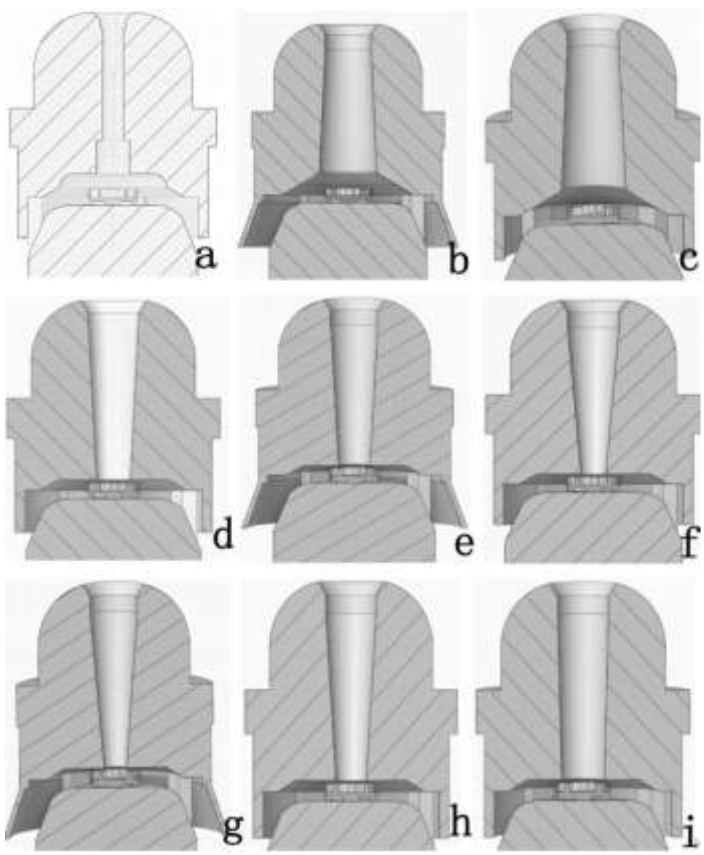

Figure 8:- Schematic Diagram of Modified Schemes.

Results Analysis:-

Pressure Contour Analysis

As shown in Figure9, which corresponds to parts a-d of Figure 8, it can be clearly observed from Figure 9a that the wind pressure of the original design's intake duct is significantly reduced compared to the external wall. In Figures 9b and 9c, the specific structural design of the U.S. military is displayed. Although this design increases the pressure inside the intake duct, the pressure difference near the wind turbine is not significant. This leads to a reduction in the driving energy received by the turbine[15], thereby reducing its torque coefficient. Figure 9d shows a new design structure. In this structure, the internal air pressure is consistently maintained at a higher level and is significantly higher than the air pressure at the turbine's outlet, resulting in a significant increase in the torque coefficient experienced by the turbine. More notably, the high-pressure area near the impeller is significantly expanded, mainly due to the reasonable adjustment of the inlet and outlet dimensions. Additionally, the low-pressure area at the inlet has also been significantly optimized. These structural and design improvements provide greater driving force for the wind turbine, thereby enhancing its rotational power generation capacity.

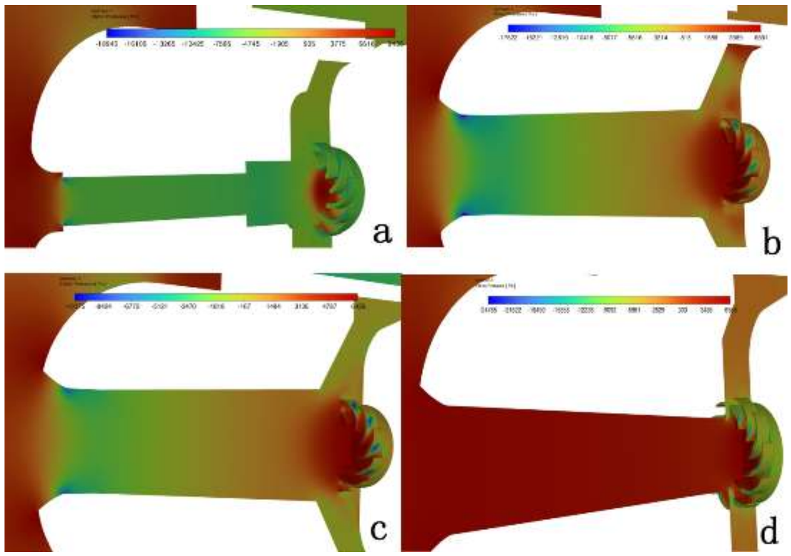

Figure9:- Overall Pressure Cloud Diagram of Inlet and Outlet Schemes.

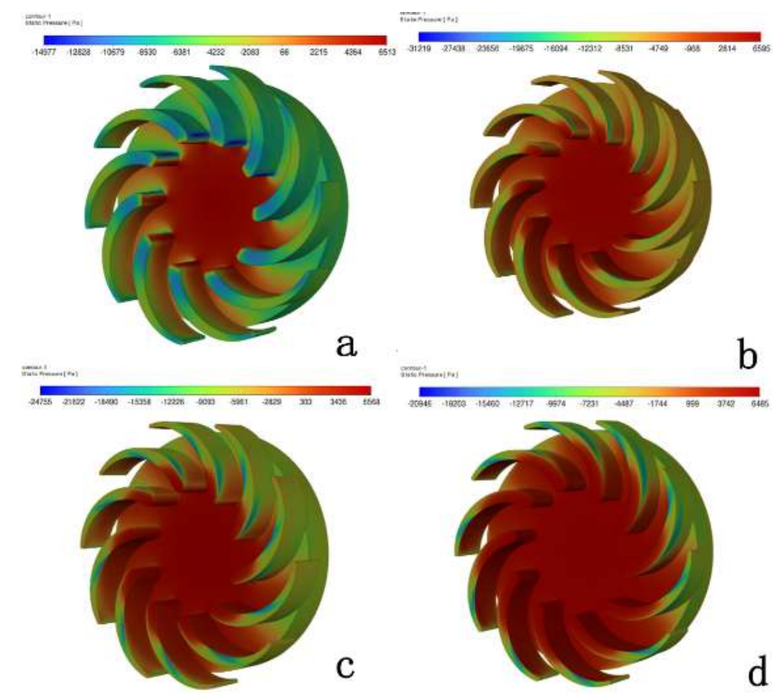

Figure 10:- Turbine Comparison under Different L.

Based on the above discussions, further studies were conducted on the pressure distribution experienced by the turbine under different tail-end diameters L. As shown in Figure 10, subfigures a-d correspond to the pressure contours of the turbine with a skirtless design under the conditions of L being R, 1.25R, 1.5R, and 1.75R, respectively. It is observed from the figures that as L gradually increases, the high-pressure area in the turbine diffuses from the center to the periphery, and correspondingly, the low-pressure area gradually diminishes. When L reaches its maximum value, the pressure on the blades is generally higher, and the pressure difference between the suction and pressure sides of the turbine decreases, leading to a reduction in the torque experienced by the turbine.

Conversely, when L is at its minimum value, a large number of low-pressure areas appear on the blades, resulting in an inability to form a significant pressure difference, thus the torque coefficient is also relatively small. Notably, in the case shown in Figure c, the distribution of high and low-pressure surfaces on the turbine is particularly evident, and these pressure areas spread orderly in the centrifugal direction of the wind turbine. This distribution pattern significantly increases the force and lever arm acting on the wind turbine, thereby increasing the torque experienced by the wind turbine, which also means that a higher torque coefficient can be obtained. This finding provides an important reference for turbine design.

Torque Analysis on the Wind Turbine

Figure 11 illustrates the torque coefficient of the turbine under different structures when the wind speed is 100m/s. It can be observed that under the structure applying the U.S. military solution, the increase in torque coefficient is not significant, which may be related to the long intake duct of that structure. However, in the subsequently improved structures, the torque coefficient begins to increase with the increase of the tail-end diameter L of the intake duct. When L > 1.5R, the torque coefficient starts to decrease with the increase of L, reaching a maximum value of 0.6 at L = 1.5R. Furthermore, it can be observed from the figure that, when the tail-end diameter of the intake duct is the same, the turbine without a skirt structure obtains a greater torque than the turbine with a skirt structure.

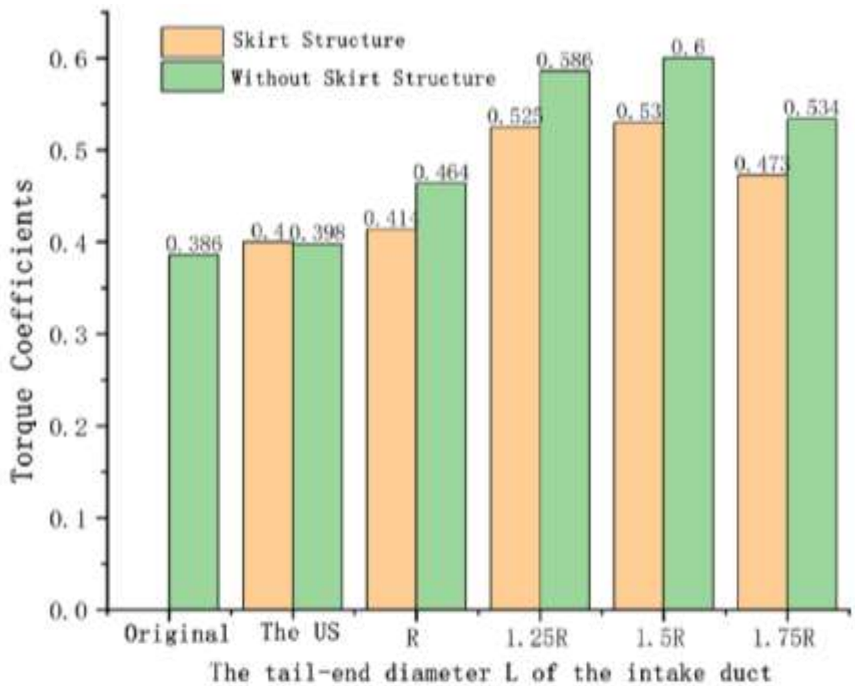

Figure 11:- Torque Coefficients under Various Conditions.

Conclusion:-

This paper, by employing polyhedral meshes and optimizing the inlet and outlet air ducts for wind turbine power generation as an example, studied the improvements of the original long intake duct design by combining the U.S. military's direct intake duct solution for fuze turbine generators with new design schemes, and mainly reached the following conclusions:

1.By reasonably adjusting the dimensions of the inlet and outlet, the high-pressure area near the impeller was successfully expanded, thereby enhancing the rotational capability of the wind turbine.

2.By adjusting the tail-end diameter of the intake duct, it was found that the working efficiency is maximized when its size is approximately 1.5 times the radius of the turbine.

3.Turbines with a skirtless structure obtained a higher torque coefficient compared to those with a skirt structure at the same tail-end diameter of the intake duct. This indicates that the skirt structure introduced some negative effects, limiting performance. Therefore, when designing and optimizing turbine structures, careful consideration should be given to whether to adopt a skirt structure to achieve better performance.

References:-

[1] 徐长江,张河.基于三维流场数值计算的引信用侧进气涡轮发电机的设计研究[J].中国电机工程学报,2006(15):144-149.

[2] 薛维清 ,邵伟春.适合实战需要的低成本涡轮交流发电机[J].现代引信,1986(01):42-55.

[3] 徐长江,张河.引信用侧进气涡轮电机三维流场数值模拟[J].弹箭与制导学报,2006(01):110-113.

[4] 陈志敏,肖志祥.风能涡轮引信电机吹风装置的轴对称收缩段设计研究[J].机械科学与技术,2001(03):397-398+314.

[5] Shen D,Zhang H, Li H. A delay arming technical scheme for underwater rocket fuze based on external conformal turbine[C]//2012.International Conference on Computer Application and System Modeling. Atlantis Press, 2012: 552-555.

[6] Sheng-gang X U, Yan-xuan W U, Ning-jun F A N, et al. Flow Field Analysis in The Domain of Work of Turbine Applied to Submunition Fuze[C]//1st International Conference on Mechanical Engineering and Material Science (MEMS 2012). Atlantis Press, 2012: 735-738.

[7] 李艾挺.叶轮机械流场数值模拟及反问题设计研究[D].中国科学院大学(中国科学院工程热物理研究所)DOI:10.27540/d.cnki.ggrws.2020.000014.

[8] 刘永. 涡轮式空气动力保险机构在迫击炮弹引信上的应用研究[D].南京理工大学,2009.

[9] 朱继南,林桂卿.引信涡轮发电机弹道发电模型和计算[J].兵工学报,1993(03):31-37.

[10] 胡坤.ANSYS FLURNT实例详解[M].北京：机械工业出版社，2018

[11] 徐长江.引信侧进气涡轮发电机气动优化研究[D].南京理工大学,2007.

[12] 曾宇,汪洪波,孙明波等.SST湍流模型改进研究综述[J].航空学报,2023,44(09):103-134.

[13] 万科,杨小贺,丁建国.风扇/增压级端区流动优化设计研究[J].燃气轮机技术,2018,31(04):53-58.DOI:10.16120/j.cnki.issn1009-2889.2018.04.009.

[14] Carl , et al. Present Capability of Ram Air-Driven Alternators Developed at HDL asFuze Power Supplies [R]. AD-A132005，1983.

[15] 刘锡明,周静,徐海梅等.水力掏槽期间突出能量及冲击波压力差分析[J].山西建筑,2014,40(29):66-67.DOI:10.13719/j.cnki.cn14-1279/tu.2014.29.039.