CALCULATION AND DESIGN OF A PODJET-TYPE HANGING WATERJET NOZZLE (SIZE 330)

NGHIÊN CỨU TÍNH TOÁN VÀ THIẾT KẾ ỐNG PHỤT KIỂU TREO PODJET SIZE 330

Hoang Van Nam*, **Pham Tien Dung**
Military Ship Design Institute, General Department of Defence Industry
**Email: nam.mt.vtktqs@gmail.com*

ABSTRACT

This paper presents the calculation and design of a Podjet-type hanging waterjet nozzle (size 330) for amphibious vehicle propulsion systems. A three-dimensional model was developed using SolidWorks and analyzed using CFD with SolidWorks Flow Simulation. The simulation is based on the Navier–Stokes equations combined with the integrated two-equation turbulence model to describe the flow inside the nozzle. The results indicate that the maximum thrust reaches approximately 7.8 kN, with a deviation of about 10% compared with the reference data of the MJP330 system. This study not only confirms the mastery of waterjet propulsion design technology within the domestic defense industry but also provides a scientific foundation for optimizing the performance of specialized marine vehicles in the future.

Keywords: *Waterjet nozzle; Podjet 330; CFD; SolidWorks Flow Simulation; Hydrodynamics.*

TÓM TẮT

Bài báo trình bày nghiên cứu tính toán và thiết kế ống phụt kiểu treo Podjet size 330 dùng cho hệ thống dẫn tiến phản lực nước của phương tiện lội nước. Mô hình ba chiều được xây dựng bằng SolidWorks và phân tích đặc tính thủy động lực học bằng phương pháp mô phỏng CFD trong SolidWorks Flow Simulation. Mô phỏng dựa trên bộ giải Navier–Stokes kết hợp mô hình rối hai phương trình tích hợp để mô tả dòng chảy trong ống phụt. Kết quả cho thấy lực đẩy cực đại đạt khoảng 7,8 kN, với sai lệch trong khoảng 10% so với dữ liệu tham chiếu của hệ thống MJP330. Nghiên cứu này không chỉ khẳng định khả năng làm chủ công nghệ thiết kế hệ thống dẫn tiến phản lực nước trong nước mà còn cung cấp cơ sở khoa học để tối ưu hóa hiệu suất cho các phương tiện thủy đặc chủng trong tương lai.

Từ khóa: *Ống phụt; Podjet 330; CFD; SolidWorks Flow Simulation; Thủy động lực học.* ☞

1. INTRODUCTION

Waterjet propulsion systems are widely used in high-speed vessels and amphibious vehicles due to their advantages in maneuverability, safety, shallow-draft operation, and high propulsion efficiency [2]. In such systems, the nozzle plays a crucial role in converting the hydraulic energy of the flow into axial thrust, directly affecting the overall propulsion performance [2, 3].

Globally, several manufacturers have developed high-performance waterjet systems, among which Marine Jet Power (Sweden) is a prominent supplier. The Podjet 330 system is commonly applied in amphibious vehicles and small high-speed crafts, featuring an impeller diameter of approximately 0.33 m, power up to 90 kW, and a rated rotational speed of 2088 rpm [4, 6]. However, imported systems are associated with high procurement costs, limited accessibility to design data, and challenges in maintenance and replacement under domestic operating conditions.

In recent years, computational fluid dynamics (CFD) has been widely applied to investigate and optimize waterjet propulsion systems. Chen et al. [7] conducted numerical studies on propulsion performance using CFD, while Kim et al. [8] analyzed hydrodynamic characteristics under various operating conditions. More recently, Li et al. [9] focused on optimizing pump and nozzle geometry to improve efficiency. These studies demonstrate the effectiveness of CFD as a powerful tool for analyzing complex flow behavior in waterjet systems.

However, most existing studies are performed under generalized or idealized conditions and do not fully account for the specific boundary conditions associated with amphibious military vehicles, such as confined installation space, varying inlet flow conditions, and operation under low-speed or bollard pull regimes. Therefore, the calculation, design, and simulation of a waterjet nozzle tailored to domestic application conditions remain a challenging and relatively unexplored problem.

This study focuses on the calculation and design of a Podjet-type hanging waterjet nozzle (size 330), combining theoretical analysis and CFD simulation using SolidWorks Flow Simulation. The objective is to evaluate the hydrodynamic performance of the nozzle under both static and dynamic operating conditions, thereby providing a scientific basis for domestic design, performance optimization, and technological autonomy in waterjet propulsion systems.

2. THEORETICAL BACKGROUND

The Podjet-type hanging waterjet nozzle (size 330) operates based on the principle of waterjet propulsion. Water is drawn into the system through the intake and accelerated by an impeller installed inside the nozzle. The high-velocity jet is then discharged through the outlet, generating a reaction force along the axial direction, which produces thrust to propel the vehicle forward.

To establish a theoretical framework for preliminary calculations, several simplifying assumptions are adopted. The flow is considered steady and incompressible, which is appropriate for water flow at low Mach numbers. The flow field is assumed to be predominantly one-dimensional along the nozzle axis, with uniform velocity distribution at the outlet section. Fluid properties such as density and viscosity are treated as constants. In addition, viscous effects, turbulence, and energy dissipation inside the nozzle are neglected, and surface roughness as well as wall friction losses are not taken into account. The influence of the impeller is represented indirectly through empirical parameters rather than explicitly modeling blade geometry. Under bollard pull conditions (static operation), the inlet velocity is assumed to be negligible compared to the outlet jet velocity, i.e., $V_{in} \approx 0$.

These assumptions allow the derivation of a simplified analytical model for estimating the hydrodynamic performance of the waterjet system. However, they also limit the accuracy of the results, as important phenomena such as turbulence, viscous losses, and possible flow separation are not considered. Therefore, the theoretical model is mainly used for preliminary estimation and trend analysis, while more accurate evaluation requires the application of Computational Fluid Dynamics (CFD).

Figure 1. Coaxial-type waterjet propulsion system developed by Marine Jet Power for military applications (adapted from Marine Jet Power [4]).

The thrust of a waterjet propulsion system is determined based on the momentum equation:

$$F = \dot{m}(V_{out} - V_{in}) \quad (1)$$

Where:

F – Thrust force (N);
$\dot{m}$ – Mass flow rate (kg/s);
V_{out} – Jet velocity at the outlet (m/s);
V_{in} – Inlet flow velocity (m/s).

At bollard pull condition, the inlet velocity is negligible compared to the jet velocity; therefore, the thrust is primarily governed by the outlet jet momentum.

3. METHODOLOGY

3.1. Theoretical calculation

The main design parameters of the waterjet system are selected based on reference data from the MJP330 nozzle.

Table 1. Main design parameters of the nozzle

Parameter	Symbol	Value	Remark
Impeller diameter	D	0.33 m	Design value
Gear ratio	i	1.16	Bevel gear transmission
Water density	ρ	1000 kg/m³	Fresh water
Nozzle outlet area	S	0.037 m²	Design value
Flow coefficient	φ	0.07	See explanation below
Impeller rotational speed	nimp	0-2088	Revolutions per minute

According to Fedotchev [1] and Kulikov [2], for single-stage axial pumps used in waterjet propulsion systems, the flow coefficient φ typically ranges from 0.05 to 0.10. In this study, a value of φ = 0.07 is adopted. The flow inside the nozzle is described using fundamental equations of fluid dynamics, including the continuity equation, Bernoulli equation, and Navier–Stokes equations. Conservation laws of mass and momentum are applied following classical fluid mechanics theory [2], [3].

The outlet jet velocity is estimated as:

$$V_{out} = \pi.D.\frac{2\pi n imp}{60}.\varphi \quad (3)$$

Under bollard pull condition, the inlet velocity is negligible, thus:

$$V_{in} = 0$$

The mass flow rate is determined from the continuity equation:

$$\dot{m} = \rho.S.V_{out} \quad (4)$$

The required power is expressed as:

$$W_t = F.V_{out} \quad (5)$$

Table 2. Theoretical calculation results of nozzle performance parameters

Engine speed (rpm)	**Impeller speed (rpm)**	**Jet velocity V_{out} (m/s)**	**Flow rate Q (m³/s)**	**Thrust F (N)**	**Power W_t (kW)**
0	0	0.00	0.000	0	0.0
600	696	5.29	0.196	1034	5.5
900	1044	7.93	0.293	2323	18.4
1200	1392	10.58	0.391	4137	43.8
1500	1740	13.22	0.489	6468	85.6
1800	2088	15.87	0.587	9316	147.8

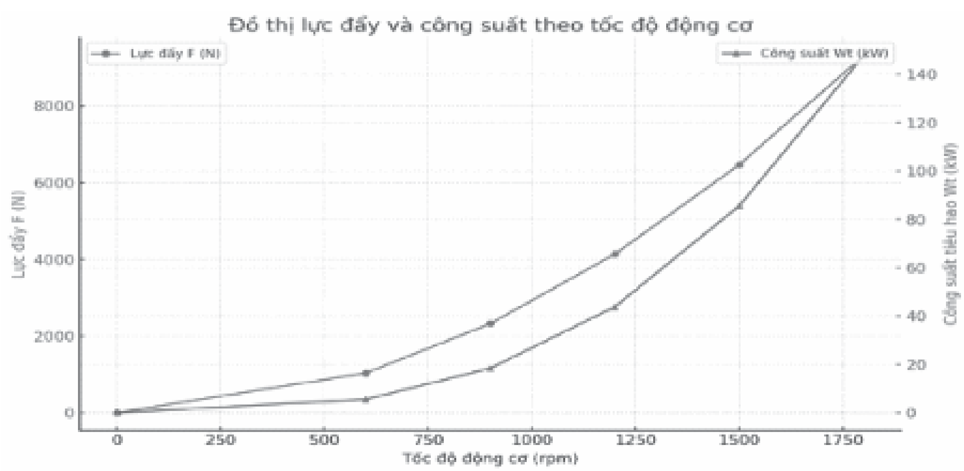

Figure 2. Theoretical thrust as a function of rotational speed under bollard pull condition.

*** Results of theoretical calculation and discussion:**

At the maximum rotational speed of 2088 rpm, the theoretical calculation yields a jet velocity of approximately 15.87 m/s, a thrust of about 9316 N, and a required power of approximately 147.8 kW. These values are significantly higher than the reference data of the MJP330 system, which reports a maximum thrust of approximately 8230 N and a maximum power of about 90 kW, indicating a clear overestimation of the system performance.

This discrepancy can be attributed to several simplifications in the theoretical model, including the neglect of hydraulic losses within the nozzle, the assumption of ideal flow conditions without viscous dissipation, and the absence of turbulence and flow separation effects. As a result, the theoretical approach provides only a preliminary estimation and does not achieve sufficient accuracy for performance evaluation. Therefore, more advanced methods, such as CFD simulation, are required to obtain more reliable predictions of the hydrodynamic performance of the waterjet system.

3.2. CFD simulation

A three-dimensional geometric model of the waterjet system, including the nozzle, impeller, and inlet/outlet flow domains, was developed using SolidWorks. Computational fluid dynamics (CFD) simulations were performed using SolidWorks Flow Simulation to evaluate the hydrodynamic characteristics of the flow inside the nozzle. The governing equations are based on the Reynolds-averaged Navier–Stokes (RANS) formulation, coupled with a two-equation turbulence model to describe turbulent flow behavior. Default turbulence parameters provided by the software were applied. Boundary conditions were defined as follows: total pressure at the inlet, static pressure at the outlet (set to zero gauge pressure), and no-slip conditions on all solid walls.

The computational domain was discretized using approximately 200,000 control volumes. The simulation was iterated until convergence was achieved, as indicated by the stabilization of monitored goal parameters, including thrust and pressure.

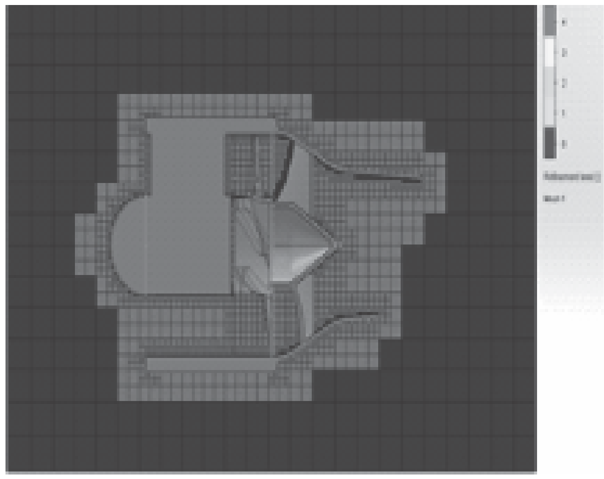

Figure 3. Meshed model of the Podjet 330 waterjet nozzle for CFD simulation.

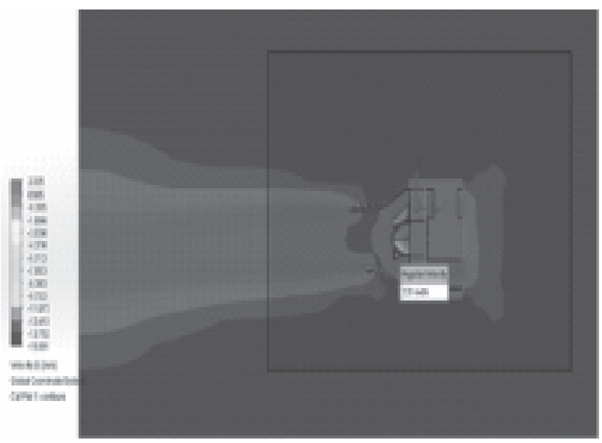

Figure 4. Velocity field distribution within the waterjet nozzle at an impeller rotational speed of n = 2088 rpm.

4. RESULTS AND DISCUSSION

4.1. CFD results under bollard pull condition ($V_s = 0$)

The CFD results indicate that the thrust increases approximately linearly with the impeller rotational speed. As the rotational speed ω increases from 0 to 2088 rpm, the thrust reaches a maximum value of approximately 7.869 kN.

Table 3. Comparison between CFD results and manufacturer data (MJP330)

Parameter	CFD prediction	Manufacturer data	Deviation (%)
Maximum thrust (kN)	7.869	8.229	4.3
Shaft power (kW)	57.4	61.4	6.5

Table 4. CFD results of thrust and power at bollard pull condition

CFD simulation results					
V_{out} (m/s)	ω (rpm)	F (N)	W_u (kW)	W_t (kW)	W_t (HP)
14.6	2088	7869	57.4	90	120
13.1	1900	6324	41.3	69	90
12.4	1800	5689	35.3	57	76
11.3	1680	4864	27.9	45	60
7.6	1150	2164	8.3	15	20

Table 5. Manufacturer data of the MJP330 system

Manufacturer data of the MJP330 waterjet system					
V_{out} (m/s)	ω (rpm)	F (N)	W_u (kW)	W_t (kW)	W_t (HP)
14.9	2088	8229	61.4	90	120.0
13.9	1900	7117	49.4	69	90.0
13.0	1800	6228	40.4	57	76.0
12.0	1680	5338	32.1	45	60.0
8.5	1150	2669	11.3	15	20.0

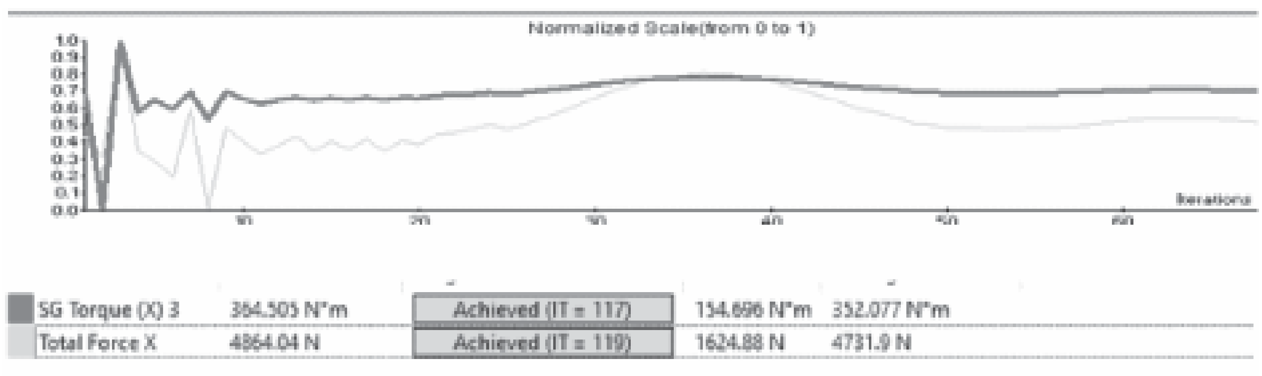

Figure 5. Flow field results of the waterjet nozzle obtained from CFD simulation at a shaft power of 60 HP.

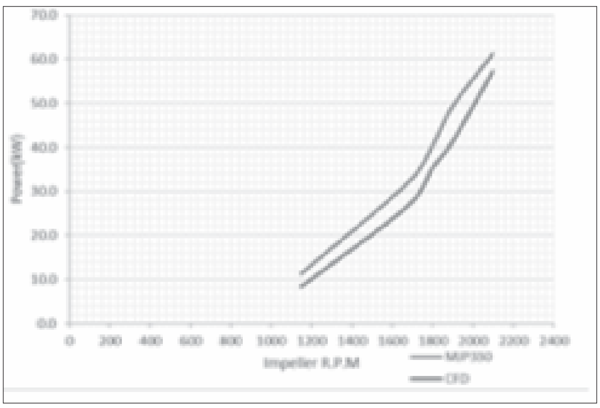

Figure 6. Comparison of useful power between the MJP330 nozzle and CFD simulation results.

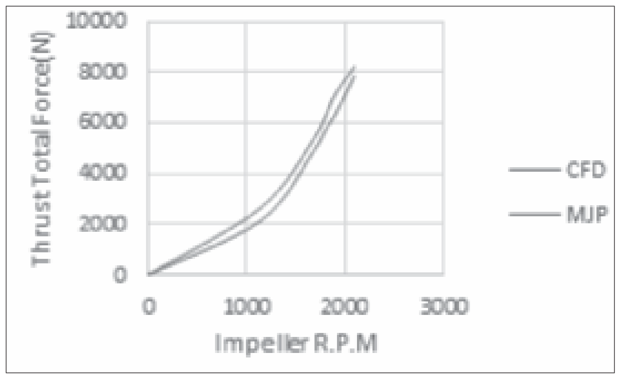

Figure 7. Thrust as a function of impeller rotational speed.

4.2. CFD simulation results under forward speed conditions

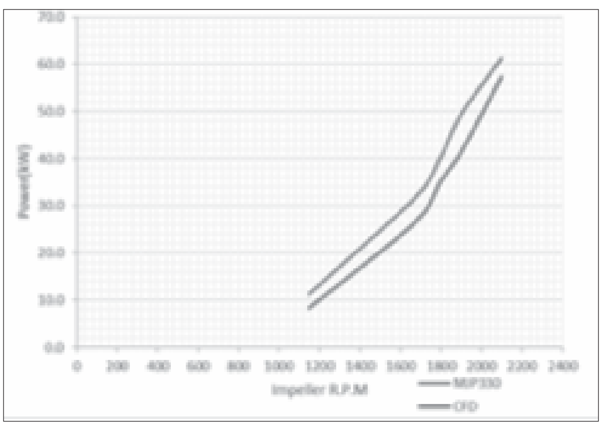

Figure 8. Comparison of useful power between the MJP330 nozzle and CFD simulation results.

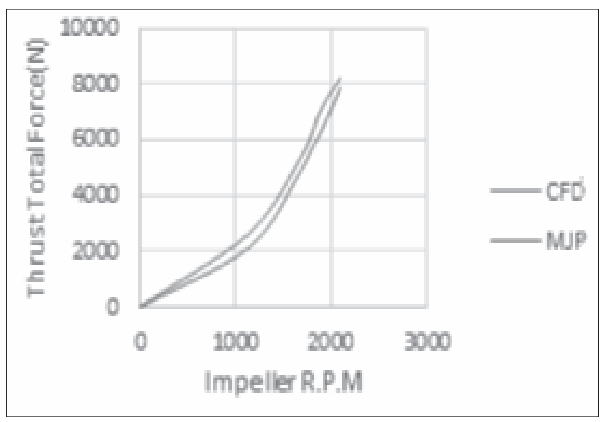

Figure 9. Thrust as a function of impeller rotational speed.

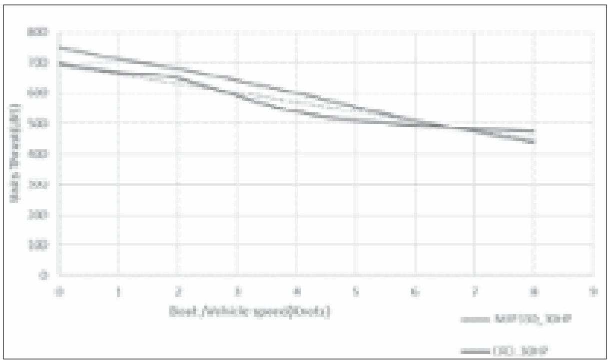

Figure 10. Thrust as a function of vehicle speed at a power of 30 HP.

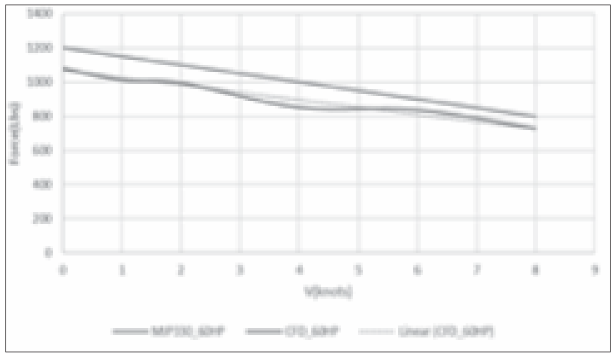

Figure 11. Variation of thrust with vehicle speed at a shaft power of 60 HP.

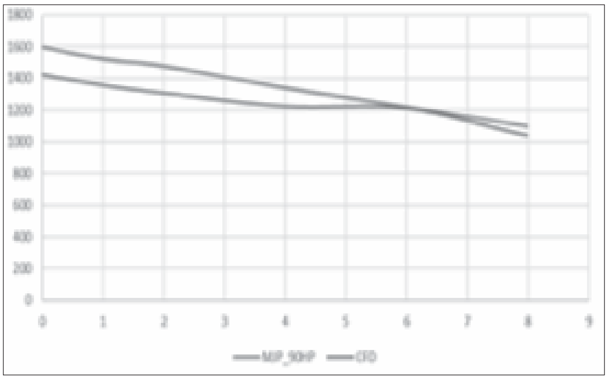

Figure 12. Variation of thrust with vehicle speed at a shaft power of 90 HP.

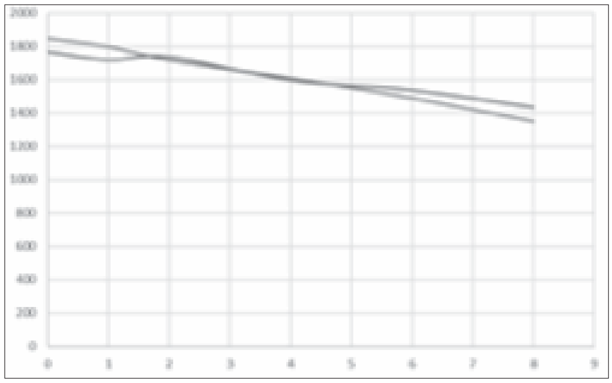

Figure 13. Variation of thrust with vehicle speed at a shaft power of 120 HP.

Table 6. Comparison of thrust between CFD simulation results and MJP330 data.

Power range	**30 HP**	**60 HP**	**90 HP**	**120 HP**
Vehicle velocity (Knots)	**$H = F_{CFD}/F_{MJP}$**			
0	93%	91%	89%	95%
1	94%	88%	89%	96%
2	95%	91%	88%	101%
4	90%	85%	91%	99%
6	97%	93%	99%	103%
8	94%	91%	94%	106%

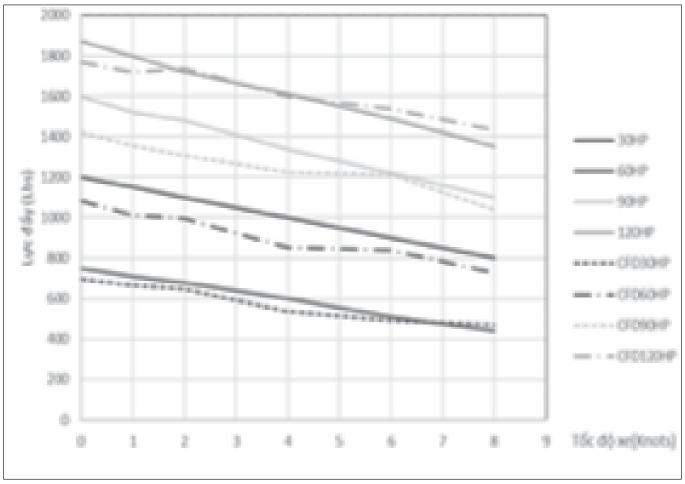

Figure 14. Power and thrust variation ranges with respect to vehicle speed.

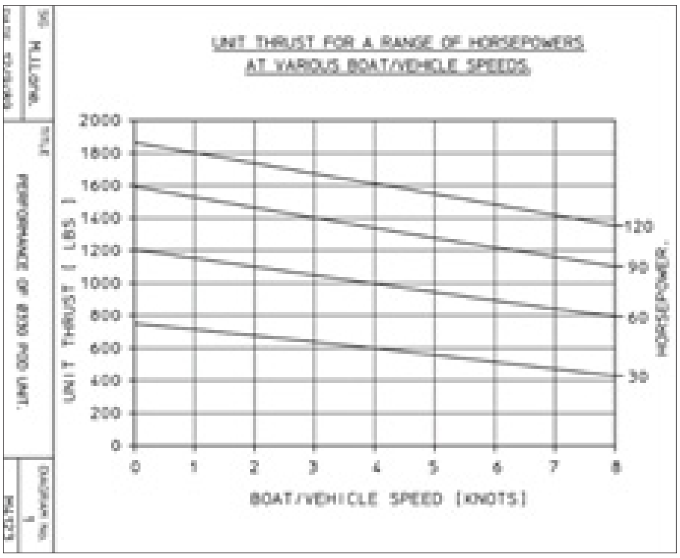

Figure 15. Thrust as a function of vehicle speed at different power levels based on manufacturer data.

The results under forward speed conditions show that the thrust decreases with increasing vehicle velocity due to the reduction in momentum difference between inlet and outlet flow. At higher power levels, the system maintains higher thrust over a wider speed range, indicating improved propulsion performance. The comparison with manufacturer data shows acceptable agreement, with deviations generally within 10%, confirming the validity of the CFD model under dynamic operating conditions.

Analysis of Discrepancies:

- CFD vs. Reality (4.3%): This may result from minor geometric differences between the simulated impeller and the actual physical model, or turbulence model constants that are not perfectly optimized for extremely high-velocity flows.

- Impact of Forward Speed: As vehicle speed increases, thrust gradually decreases ☞

due to the narrowing momentum difference between the inlet and outlet flows (Figures 10-13). At high power (120 HP), the system maintains more stable thrust across a wider speed range, indicating the design's robust dynamic efficiency.

5. CONCLUSIONS

This study developed a design and evaluation methodology for a Podjet-type hanging waterjet nozzle (size 330) based on a combination of theoretical analysis and CFD simulation. The CFD results obtained using SolidWorks Flow Simulation indicate that the maximum thrust reaches approximately 7.8 kN, with a deviation of about 10% compared to the reference data of the MJP330 system. This demonstrates that the numerical model provides reliable predictions of the hydrodynamic performance of the waterjet propulsion system. The findings confirm the feasibility of domestic design and manufacturing of a waterjet nozzle equivalent to the Podjet 330 system. Furthermore, the results provide a foundation for future studies on geometric optimization and experimental validation.

Acknowledgement:

This research was supported by the scientific and technological funding program of the General Department of Defence Industry, Ministry of National Defence.❖

References:

[1]. Fedotchev VA, “*Optimal Design Methodology and Research on Axial Pump Impellers*”. PhD Thesis, Moscow, 2005.

[2]. Kulikov SV, Khramkin MF, “*Water-Jet Propulsion: Theory and Calculation*”. Sudostroenie, 1980.

[3]. Stepanov AI, “*Centrifugal and Axial Flow Pumps*”. State Scientific Publishing, 1960.

[4]. Marine Jet Power AB, “*MJP 330 Pod – Installation & Maintenance Manual*”, 2015.

[5]. Russian River Register, P.013-2006, “*Движительно-рулевые и подруливающие устройства*”, 2006.

[6]. Marine Jet Power AB (Sweden), “*Podjet Waterjet Series 330 – Technical Manual*”, 2018.

[7]. Chen X., Wang Y., Liu H., Zhang Z, “*Numerical investigation of waterjet propulsion performance using CFD*”. Ocean Engineering, 2020, 209:107456.

[8]. Kim J., Park S., Lee D., Choi H, “*Hydrodynamic performance analysis of waterjet propulsion systems under different operating conditions*”. Applied Ocean Research, 2021, 110:102590.

[9]. Li Y., Zhao X., Sun Q., Chen L, “*CFD-based optimization of waterjet pump and nozzle geometry*”. Journal of Marine Science and Engineering, 2022, 10(4):512.