Hydrodynamic Development of Azimuthing Podded Propulsion System*1

By Isao FUNENO (Member)*2

Hydrodynamic development of the azimuthing podded propulsion system is presented. First in process of the development, the geometry of the pod housing and strut has been optimized by means of numerical simulation technique based on computational fluid dynamics (CFD) solving the Navier-Stokes’ equations. In the numerical simulation, there are the steady analysis by means of the sink-disc method as effects of a propeller race simply and the quasi-steady analysis in order to investigate accurately hydrodynamic interaction between the full propeller and the pod housing in straight forward condition, that is, non-azimuthing condition. In that time the effects of fins attached the pod were studied. Furthermore the unsteady analysis has been performed in order to investigate exactly effects of the unsteady terms in related equations. Especially the unsteady analysis is essential for investigation of the flow around the podded propulsion system in oblique flow, that is, steering condition. The results of the investigation about the unsteady hydrodynamic forces acting on the whole unit in neutral and oblique rudder angles are shown and discussed in detail. By using these analytical methods, the optimization of the pod and strut was accomplished successfully by compromise with the dimension of the built-in motor as to decrease the drag of the pod housing and strut. Next open water tests and cavitation tests were conducted with the final pod shape. Based on the results of the tank tests, the computed results are verified and basic knowledge concerned with the propulsion performance was obtained. As a result, the author can find that the computed results agree very well with the measured ones and the final geometry of the pod housing and strut has excellent performance.

Keywords: Podded Propulsion, Propeller, Propulsion Performance, Unsteady Analysis, CFD, Steering Performance

1. Introduction

Recently the electric podded propulsion has become the standard for large cruise vessels, icebreakers and chemical tankers owing to the increase of required inboard electric power and generator capacity for electrically powered cargo gears. In this propulsion system, an electric propulsion motor is located inside a pod connected by a strut with stern overhang of a hull and a fixed pitch propeller is driven directly in front and/or tail of the pod.3

This propulsion system provides the following various advantages,

- By disuse of main engines connected with line shafts, the degrees of freedom of the stern hull design are raised. Therefore the propulsion performance is improved by the buttock flow stern shape that provides lower hull resistance, especially in full hulls reduces harmful longitudinal vortex,
- As the system makes it possible to equip even after lunching, it enables to reduce costs in construction,
- Maneuverability in harbor is remarkably improved provided that the system is equipped with a 360 degrees azimuthing gear.

Under the above-mentioned background, Kawasaki Heavy Industries (KHI) has recently developed the azimuthing podded propulsion system and completed the prototype4 (refer Fig.1).

In this paper, firstly the author describes hydrodynamic development of the KHI podded propulsion system. In the process of development, the geometry of the pod and the strut has been optimized by means of numerical simulation technique based on Computational Fluid Dynamics (CFD). In this numerical simulation, there are steady analysis based on the sink-disc method as effects of propeller race,
quasi-steady analysis and unsteady analysis in order to investigate accurately interaction between the propeller and the pod and strut. Especially the unsteady analysis is essential for investigation of the flow around the podded propulsion system in oblique flow, that is, steering condition. Results of investigation about unsteady hydrodynamic forces acting on the whole unit in neutral and oblique rudder angles are shown.

Next, open water tests and cavitation tests were conducted with the final pod shape. Based on the results of the tank tests, computed results are verified and basic knowledge concerned with propulsion performance is shown.

Lastly discussion and conclusion are described.

2. Computational Methods

The Reynolds Averaged Navier-Stokes' Equations (RANSE) for incompressible fluid are applied to analyze the viscous flow around the podded propulsion. The numerical procedures to solve the RANSE are described in the author's previous papers on the steady analysis\textsuperscript{[5-6]} and the unsteady analysis\textsuperscript{[7]}. The summary of the procedures is as follows. Firstly the RANSE are discretized based on the finite volume method. As the discretization method of the convection terms of the RANSE, MAR scheme that is a kind of TVD scheme is adopted in order to reduce the numerical viscosity as possible. The effects of turbulence are modeled by the RNG k-\(\varepsilon\) model and the wall function boundary conditions are used. Next the discretized equations are computed numerically based on the SIMPLE algorithm for the steady analysis and the PISO algorithm with the fully implicit scheme for the unsteady analysis.

The steady analysis is applied to optimize the pod and strut shapes. Also the optimization is carried out in straight course condition. In this analysis, implementation of a rotating propeller is based on the sink-disc method expediently. The body forces are uniformly distributed on the propeller disc so as to be equivalent to the specified propeller thrust and torque. Thus the hydrodynamic interaction between the propeller, pod and strut is ignored.

Next the quasi-steady analysis and the unsteady analysis are applied to analyze the flow around the final pod shape with the full geometry of the propeller in straight and oblique flow. In the quasi-steady analysis, the rotation of propeller is froze at a circumferential position so as to analyze the flow around the propeller and the pod and strut. This analysis is able to take hydrodynamic interaction between the propeller and the pod and strut into account accurately except the effect of the unsteady terms. But the unsteady analysis is able to take the interaction and the effect of the unsteady terms into account exactly by means of the sliding mesh technique\textsuperscript{[8]}. The solution methods are based on the commercial CFD software STAR-CD\textsuperscript{[9]}. The software has the implementation of the above-mentioned all functions and has been tested intensively on versatility and accuracy of the propeller flow computation. Also in order to reduce CPU time needed to obtain a solution, all the computations were carried out by using the parallel computing technique, which each computational domain was assigned to each CPU.

The grid generation is based on the unstructured grid technique. By this technique, the computational domain around such a complex geometry as the podded propulsion unit is divided optimally into many blocks and the proper grid arrangement on computational accuracy and efficiency (convergence ratio) can be generated at each block. The manners for the grid generation followed the guidelines in the reference\textsuperscript{[9]}. In practice, after the geometry of the
propeller and the pod and strut were modeled by using a CAD software, the grid generation was carried out by the commercial software GRIDGEN\(^7\). Figs. 2 and 3 show the grids on the surface of the initial geometry of the pod and strut with the propeller sink-disc and the grids on the surface of the final whole unit at the quasi-steady analysis respectively. The numerical grids constructed about 500,000 cells for the steady analysis, about 1,020,000 cells for the quasi-steady analysis and about 1,200,000 cells for the unsteady analysis.

3. Optimization of Pod Shape

3.1 Objective

The objective of optimization is to develop the efficiently optimal geometry of the pod and strut in consideration of the dimensions of the built-in motor and the ventilation flue of the motor cooling air. Thus by means of the numerical simulation based on CFD solving the RANSE, hydrodynamically unfavorable flow around the pod and strut with a rotating propeller is investigated and then improved to the proper geometry.

3.2 Steady Analysis

The computation cases listed in Table 1 were carried out in order to optimize the pod and strut shape.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>(D_{pod}/D_p)</th>
<th>Section of Strut</th>
<th>Pod tail</th>
<th>Fins</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.47</td>
<td>Oval section</td>
<td>Cone</td>
<td>Nil</td>
</tr>
<tr>
<td>2</td>
<td>0.51</td>
<td>Oval section</td>
<td>Cone</td>
<td>Nil</td>
</tr>
<tr>
<td>3</td>
<td>0.51</td>
<td>Wing section-1</td>
<td>Cone</td>
<td>Nil</td>
</tr>
<tr>
<td>4</td>
<td>0.51</td>
<td>Wing section-1</td>
<td>Truncated</td>
<td>Nil</td>
</tr>
<tr>
<td>5</td>
<td>0.45</td>
<td>Wing section-1</td>
<td>Cone</td>
<td>Nil</td>
</tr>
<tr>
<td>6</td>
<td>0.46</td>
<td>Wing section-2</td>
<td>Cone</td>
<td>Nil</td>
</tr>
<tr>
<td>7</td>
<td>0.46</td>
<td>Wing section-2</td>
<td>Cone</td>
<td>One Bottom Fin</td>
</tr>
<tr>
<td>8</td>
<td>0.46</td>
<td>Wing section-2</td>
<td>Cone</td>
<td>Three Tail Fins</td>
</tr>
</tbody>
</table>

Where, \(D_{pod}\) diameter of pod, \(D_p\) diameter of propeller.

From these computations, the author can find the followings. The smaller diameter ratio of the pod to propeller: \(D_{pod}/D_p\) decreases the resistance of the pod. For reference, however, according to Yamano' investigation on Kawasaki rudder bulb\(^8\) the diameter ratio around 0.35 is optimal. Nevertheless, the diameter ratio settled finally into 0.46 by compromise with the motor dimensions. The modification of outlines of strut section precludes the excessive concentration of negative pressure at the leading edge region and the formation of vortex flow at the trailing edge region. For example, Fig. 4 shows the comparison of surface pressure between Case No.2 and Case No.3. The truncated tail end of the pod contributes little to
the decrease in resistance of the pod. The strut of the wing section with higher lift-drag ratio allows to decrease the pod resistance by around 20%. From Fig.5, the proper bottom fins cancel the undesired lateral forces: $K_{p}$ in straight course, which are caused by the swirl race of propeller. It is found that the addition of fins contributes to decrease in pod resistance. For example, from Fig.6 that shows distribution of velocity vectors and non-dimensional longitudinal vorticity: $\omega_z/(V/D_p)$, the author can find that the propeller swirl is almost diminished by the stator of the three tail fins.

Fig. 4 Pressure contours on pod and strut.

3.3 Quasi-steady Analysis

The quasi-steady analysis was carried out in regard to Case No.6 in straight course basically. Though the computations were carried out every 15 degrees of a blade position, the fluctuation of propeller bearing forces is very small. From Fig.7, the author can find that the pod resistance: $K_{p}$, by the quasi-steady analysis is almost the same as the one by the steady analysis. Consequently the steady analysis is useful for screening study of the only pod shape.

Fig. 7 Resistance of pod and strut.

3.4 Unsteady Analysis

The unsteady analysis was carried out in regard to Case No.6 like the quasi-steady analysis, provided that the advance coefficient: $J$ is 0.666 as the design point. From Fig.8 that shows axial, radial and tangential components of fluctuation forces per blade during one revolution ($K_{r1}(\theta), K_{r2}(\theta), K_{t1}(\theta)$), the author can find that the fluctuation of the forces per

Fig. 8 Fluctuation forces per blade during one revolution ($J$=0.666).
blade in straight course is very small. The fluctuation forces of the pod and strut are also very small. But once the rudder angles are set to an angle, the circumstance is changed completely. From Fig.9 that shows pressure contour on the propeller and the pod and strut, the author can estimate that the cavitation is generated on a large scale at the propeller tip region and the leading edge of the strut. Also from Fig.10, the author can find that the propeller bearing forces are very large. These circumstances have almost analogy with ones of a propeller with a raked shaft.

Fig. 9 Contour of pressure coefficient: $C_{p_{m}}$ on podded propulsion in oblique flow (rudder angle: $\delta = 30^\circ$, $J=0.666$).

4. Experiments and Verification

4.1 Objectives

The objectives of experiment are to verify the results by the numerical simulation based on CFD, to observe and evaluate the cavitation around the podded propulsion and to confirm the effect of the fins. For the purposes, the open-water tests and the cavitation observation tests were conducted at the upgraded depressurized towing tank in MARIN (The Netherlands). The tested pod shapes are Case No.6 and No.8 in Table 1. Also the proper stock propeller in MARIN was used for the model propeller of the podded propulsion.

4.2 Open-water Characteristics

Fig.11 shows the open-water characteristics of Case No.6, that is, without fins, compared with ones computed by the quasi-steady analysis. From this figure, the author can find that the computed results agree with the measured ones very well, especially the torque coefficients. Fig.12 shows the measured open-water characteristics of the propeller alone and the podded propulsion. From this figure, the author can find that the propeller thrust and torque are increased owing to the disturbance potential wake caused by the pod and strut. But the total thrust of
Fig. 13 Cavitation observation in oblique flow (rudder angle: $\delta=30^\circ$, $J=0.666$, $\sigma_{a}=1.50$).

effect, the further investigation including the improvement of the fin shapes should be continued. Though the fluctuation forces of the propeller and the pod were measured at that time, it was difficult to obtain the proper data to analyze owing to the vibration of the bevel gear of the motor. It is the subject to be tackled in the next step.

4.3 Cavitation Observation Tests

Fig.13 shows the photograph of the cavitation observation at the almost hard starboard (rudder angle: 30 degrees). The cavitation number: $\sigma_{a}$ was 1.50 which corresponds to the condition that the speed of a ferry is for example about 25 knot. The author can find that correspondence between the observed cavitation and the computed one (refer Fig.9) is very good. But there was no harmful cavitation at the design point in straight course.

5. Discussion

It is possible to optimize the geometry of the pod and strut only by the steady analysis with the propeller sink-disc technique simply. In straight course, the podded propulsion generates little propeller bearing forces from the results computed by the quasi-steady and the unsteady analysis. On the other hand, in oblique flow, that is, steering condition the podded propulsion brings about large propeller bearing forces and severe cavitation from the results.
computed by the unsteady analysis and observed by the tank tests. Accordingly in high-speed navigation the author recommends to steer by using rudders separately. Nevertheless, it is very useful for maneuvering a ship in harbor to operate the azimuthing podded propulsion system instead of stern thrusters.

6. Conclusion

The optimization of the pod and strut are successfully accomplished by using the steady analysis with the propeller sink-disc technique based on CFD solving the RANSE. Furthermore the unsteady analysis taken the full geometry of rotating propeller and the unsteady terms in account exactly is shown. Consequently the author can find that the propeller bearing forces in straight course are very small, on the other hand ones in oblique flow, that is, steering condition are very large.

From the results of the open-water tests and the cavitation observation tests of the podded propulsion unit, the author can find that the computed results agree well with the measured ones. Also the severe cavitation is observed in oblique flow as estimated by means of the unsteady analysis. But the author can find that this podded propulsion has hydrodynamically excellent propulsion performance in straight course because of no harmful cavitation. The author intends to improve further hydrodynamic performance of the podded propulsion system by using these methods in the next steps.

References

7) http://www.pointwise.com/

Discussions

【討論】（兵庫教育大学）山野惟夫
考慮すべき要素の多いボッドプロペラの形状の最適化を行う方法として、CFDを応用した方法を開発して、その有効性を実証したことに敬意を表します。
下記の点について教えて下さい:
Table 1に示されている計8ケースについて比較計算をされておりますが、本講演概要に示されている結果はその一部に過ぎません。他のケースについても、効率の比較等差支えない範囲で示して下さい。
【回答】
ご討論を有り難うございます。以下に回答申し上げます。Fig.Aにおいて、ケース1〜4はケース1をベースに、およびケース5〜8はケース5をベースとした装置全体の単独効率の比較を示します。この図より、2つのグループに分かれていますが、これは電動機出力：pが当初2700kWで計画されていましたが、諸般の事情で途中から1000kWに変更されたためです。この比較図より、ケース1から2でボッドとプロペラの直径比が大きくなると効率が悪くなり、ケース2から3で圧力分布の改善
や有害な剥離渦の除去はできましたが、ストラット断面の改良が不十分で効率が若干悪化しました。ケース3から4では円錐台形のボッド後端形状としましたが、ほとんど改善は見られませんでした。ケース5はケース1～4で得られた知見をふまえて、ケース3をベースに再設計したものです。更にケース6でストラット断面を改良しました。その結果、抵抗係数
cのベースで約20％低減できました。さらにケース7でボットムフィンを付加し、ケース8でテイルフィンを付加しました。結局ケース6をベースとして約8％の効率向上が推定されました。

【討論】（九州大学大学院）中武一明
プロペラをSink discで表しておられるので、ボッドによる伴流の影響がプロペラ性能の中に取り込まれていなければいかと思われますが、如何でしょうか。

【回答】
ご討論を有り難うございます。以下に回答申し上げます。本論でも述べておりますように、今回示しましたSink disc法では確かにプロペラとボッドおよびストラットの流力の相互干渉が厳密に考慮できません。しかし、ボッド形状の最適化の初期検討段階において、この簡易的な手法は多数のケースについて現実的検討時間内でスクリーニングするには実用的な精度を有しているものと考えております。実際、Fig.7に示しますように計算値と比較した結果、ボッド部の抵抗は十分な精度があるものと考えております。
また、最終形状には厳密な非定常計算で相互干渉影響も含めた性能確認を取っておくことが重要かと思われます。

【討論】（海上技術安全研究所）右近良孝
1. Introductionについて
（1）講演概要集でin front of the podとトラクタ型しかボッド推進がない様に記述されていますが、ブッシャ型はどういう利点と欠点があるとお考えでしょうか。
（2）ボッド船型にすると、設計自由度は無限になるのでしょうか。
（3）longitudinal vortexは単純にharmfulなのでしょうか。
2. Computational Methods
（1）MARとは何の略でどの様なスキームでしょうか。
（2）第2パラグラフでは定常計算（プロペラ吸い込みディス法と組み合わせ）プロペラとボッドなどとの干渉を無視したとあり、第3パラグラフでは非定常計算では考慮したとあり、矛盾している様に思われますが、いかがでしょうか。
3. Optimization of Pod Shape
（1）3.1で“inconvenient flow”との記述がありますが、どんな流れでしょうか。
（2）3.3でFig.7の計算と実験を合っていませんが、何故かそれぞれの理由をお示し下さい。
4. Experiments and Verification
（1）4.2でFig.11における実験と計算が合っているとの記述がありますが、効率は10％以上も異なり、合っているとは言えないと考えます。如何でしょうか。
（2）ボッドに付けたプロペラは単独時より効率が高くなっていますかなぜでしょうか。

【回答】
ご討論を有り難うございます。以下に回答申し上げます。
1-(1)ブッシャー型は、ボッドおよびストラットの後流による伴流利得が得られますが、伴流分布が不均一となるため、ベアリングフォース等が発生しやすく、振動対策上好ましくありません。過去の実績を見ますと、トラクタ型の方が多いようです。
1-(2)ご指摘有難う御座います。本論文で訂正させて頂きます。ボッド型推進装置の配置と船首形状には最適の組み合わせがあり、設計の自由度はラインシャフト方式より高いものと思われます。
1-(3)船首縦渦、その混合作用により肥大船首端に伴流利得を向上させるメリットがあると言われますが、しかし現在ではその縦渦の発生によって船体抵抗を増加させる影響方が大きいようです。まず、船体抵抗が低い船型を開発することが先決と考えております。
2-(1)MARスキュームとは、Monotone Advection and Reconstruction schemeの略で、TVDスキュームの一種であり、対流項の離散化に使われ、空間2次精度を持ちます。これにより数値粘性をなるべく抑制することができます。
2-(2)中武先生への回答をご参照下さい。
3-(1)剥離渦が発生し、流れが滑らかでないような流れは、推進効率を向上させる上で、不都合なのでこのような表現をつかいました。なお、本文ではunfavorable flowと
訂正させて頂きます。
3-(2) CFD 計算は定性的にはよく合っていると考えますが、乱流モデルを使って全計算領域で乱流として計算を実行しているのに対して、模型実験ではまだ層流域が残っていたのではないかと考えております。このような層流／乱流影響も含めて粘性に基づく尺度影響は今後の研究課題とすべきものです。
4-(1)これは、模型プロペラ翼の翼根部付近に層流領域が存在することによるものと思われます。先述しましたように計算は、全計算領域で乱流と仮定して計算しているために、このような誤差が生じたものと思われます。しかしポテンシャル理論によると、比較的近い実験的修正係数を導入すれば推定精度は向上するものと思われます。
4-(2)本文にも述べましたが、ポッドおよびストラットによる掲乱ポテンシャル伴流によりプロペラ効率が上昇したものと思われます。この現象は一般にトルクが減少しプロペラ効率比が 1 より大きくなることと同様の analogy と考えられ、今後の研究課題とさせて頂きます。

【討論】（海上技術安全研究所）久米健一
1. truncated tail end がポッドの抵抗を減ずるという結果について、摩擦抵抗の減少以上に粘性圧力抵抗の増加によって逆に抵抗は増える気がしますが、どのような考察をお持ちでしょうか？
2. プロペラそのもののデザインについて論じられていませんが、電動モータ駆動であることやポッドおよびストラットの伴流を考慮すれば、どのようなプロペラ設計が効果的だと思われでしょうか？

【回答】
ご討論を有り難うございます。以下に回答申し上げます。
1. コーン型のポッド後端では、ハブ溝と同様にプロペラ翼根部からの渦が後端部で集まし集中するために、大きな負圧領域が形成され、抵抗となります 4)。この集中を緩和するために円錐台型のポッド後端部を検討したのですが、十分に抵抗されなかった様です。
2. 今回は、時間の都合で発表しませんでしたが、既にプロペラチャートと水槽実験結果をベースにしてポッドおよびストラットとの相互干渉影響を考慮した簡易的なプロペラ設計法を開発しており、初期検討段階では実