Numerical Simulation of Rocket Engine Internal Flows

Project Representative

Nobuyuki Tsuboi   JAXA's Engineering Digital Innovation Center, Japan Aerospace Exploration Agency

Authors

Shinichi Tsuda [*1], Nobuhiro Yamanishi [*1], Chisachi Kato [*2] and Taro Shimizu [*1]

＊1 JAXA's Engineering Digital Innovation Center, Japan Aerospace Exploration Agency

＊2 Institute of Industrial Science, The University of Tokyo

In this fiscal year, we have carried out two types of simulations which are related to vibrations in rocket engine turbopumps. One is a simulation of unsteady cavitating flows in a liquid oxygen pump, and the other is that of a phenomenon of rotor-stator interaction in a liquid hydrogen pump. We have successfully simulated both cases and the simulations have shown potentials to clarify some unstable axial vibration problems which should be solved to develop more reliable rocket engines.

Keywords: rocket turbopump, large eddy simulation, cavitation, dynamic gain factors, axial vibration, rotor-stator interaction

1. Cavitation Instability Analyses in Liquid Oxygen Pump

To achieve stable operations at a high rotational speed under a low inlet pressure, rocket engine turbopumps are generally equipped with an axial-flow inducer stage. As the inlet pressure is decreased and local pressure becomes lower than the vapor pressure, cavitation gradually develops on the suction side of inducer's blades and/or near their tip and finally leads to the breakdown of the inducer performance. In addition, cavitation instabilities, such as rotating cavitation and cavitation surge, are often observed in experiments and cause serious shaft vibration and/or fluctuations of the blade stresses. Therefore, it is an important issue to understand the physical mechanism of unsteady cavitation phenomena related to internal flows of a turbopump inducer.

In the past, we have developed a Large Eddy Simulation (LES) code for accurate computations of unsteady flows in turbomachinery, and performed computations of cavitating flows for a test inducer. Our LES code solves the Navier-Stokes equations of weakly-compressible flow, in which standard or dynamic Smagorinsky model is implemented as sub-grid scale (SGS) model. The code is based on a finite element method with hexahedral elements and has the second-order accuracy both in time and space[1]. By the multi-frame of reference function based on an overset method, it is possible to compute rotor-stator interactions[2]. For computation of cavitating flows, we have used the cavitation model proposed by Okita et al.[3] In this model, the evolution of cavitation is represented by source/sink of the vapor phase.

In this fiscal year, we have carried out LES analyses of unsteady cavitating flows where flow rate or pressure at inlet is fluctuated at a certain frequency. This kind of analyses has

a potential to define a so-called "dynamic gain factors" which can be used to make judgements whether unstable cavitation occurs or not.

In this simulation, we have employed a test inducer with three helical blades. The total mesh consisted of approximately 8.5 million hexahedral elements. The main calculation conditions were as follows; inlet flow rate was 23 liters/s, rotating speed was 17,700rpm, and inlet cavitation number was $\sigma = 0.04$. Two kinds of inlet boundary conditions were imposed to calculate the dynamic gain factors. One was that flow rate was fluctuated with pressure being constant. The other was that pressure was fluctuated with flow rate being constant. We can calculate a head gain (for flow rate change) and mass flow gain factor from the former calculation, and can calculate another head gain (for pressure change) and cavitation compliance from the latter calculation. Generally, the larger mass flow gain factor makes the system unstable while the larger cavitation compliance makes them stable. Therefore, these parameters are useful to judge the cavitation instability although the evaluation is not easy in experiments, at least in the current measurement techniques. The present simulation image is shown in Fig. 1. The fluctuation frequency of flow rate or pressure at inlet is given as 4 times slower than the rotating frequency of the inducers.

Before giving fluctuation, we calculated the cavitation flow around the steady operation point. The obtained flow field is shown in Fig. 2. Although the cavitation development is insufficient, we could confirm a structure of tip vortex cavitation which is originated from a shear stress. (see Fig. 2(a).) This structure was captured by our LES analyses. Figure 3 is the calculated cavity volume and mass flow gain factors in the case of "inlet flow fluctuation" condition, and Fig. 4 is
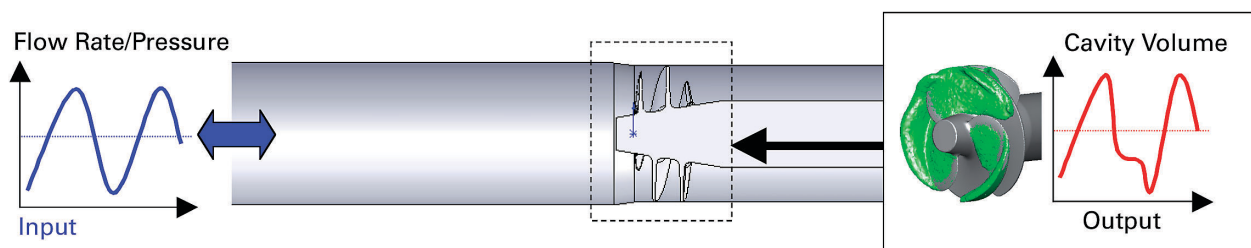
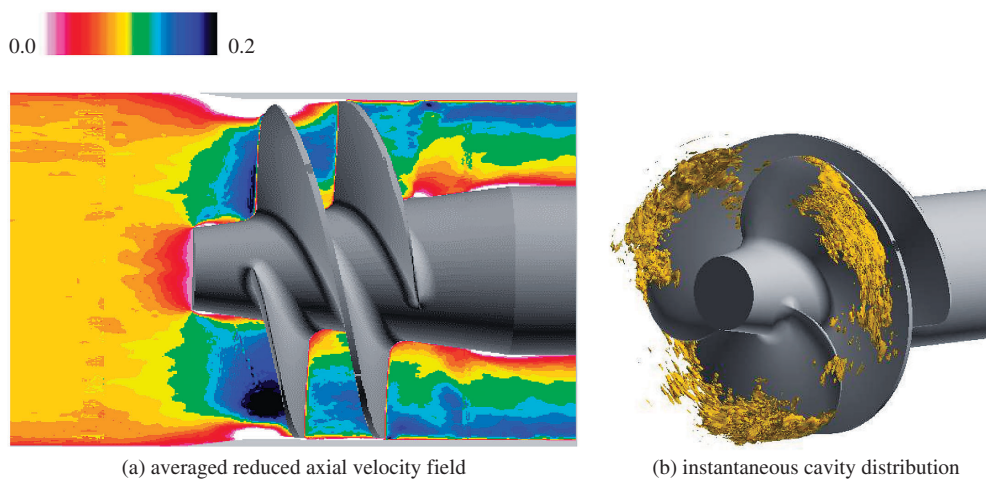Fig. 1 Schematic of the present method to calculate dynamic gain factors.

(a) averaged reduced axial velocity field

(b) instantaneous cavity distribution

Fig. 2 Flow field before giving fluctuation. (Iso-surface of the void fraction 2% is shown in (b).)

(a) Time change of reduced cavity volume

(b) Time change of mass flow gain factor
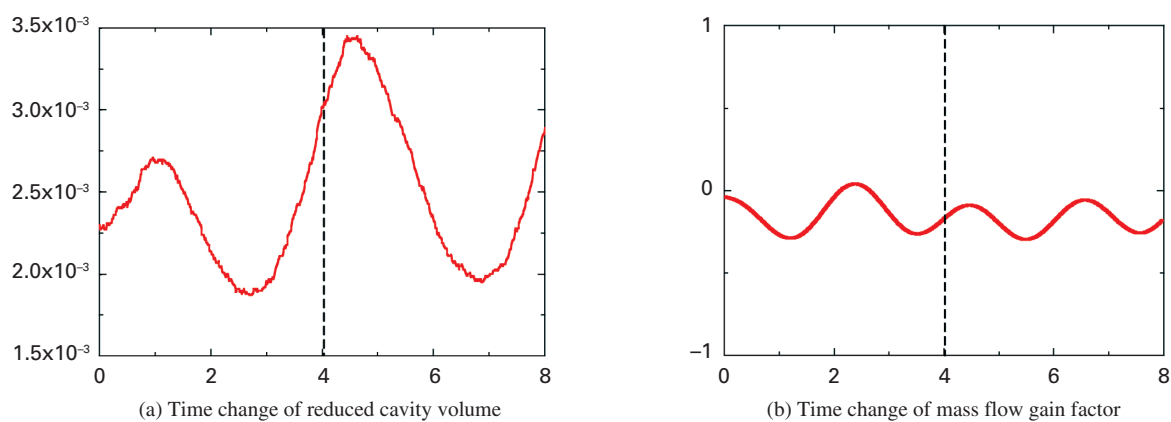
Fig. 3 Calculation results for an "inlet flow rate fluctuation" condition. (Each horizontal axis is the number of revolution.)

(a) Time change of cavitation compliance
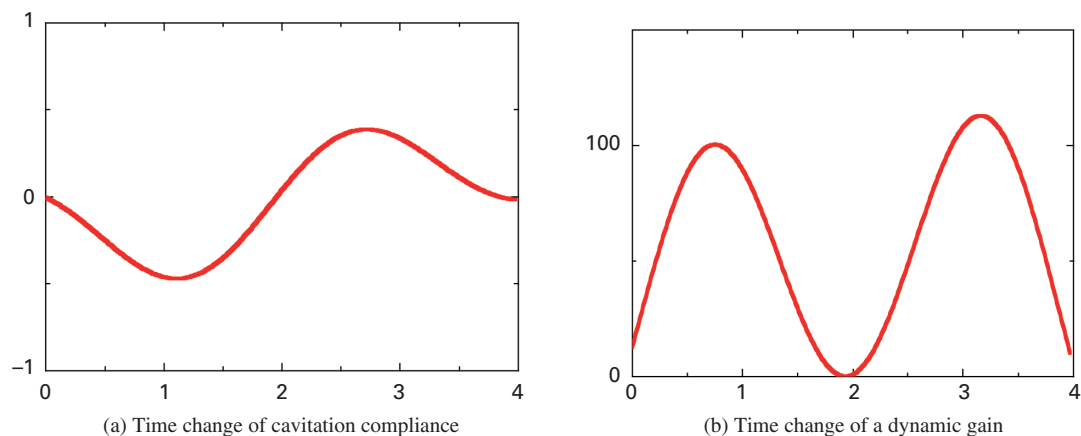
(b) Time change of a dynamic gain

Fig. 4 Calculation results for an "inlet pressure fluctuation" condition. (Each horizontal axis is the number of revolution.)

the corresponding results for the "inlet pressure fluctuation" condition. As shown in Fig. 3, we could extract the cavity volume change that was originated from the flow rate change. One of the accomplishments in these calculations is that we could directly calculate the cavity volume change and dynamic gain factors such as mass flow gain factor and cavitation compliance. To our knowledge, it is the first results where the dynamic gain factors are calculated using unsteady cavitation CFD with LES analyses. Although the validation of the calculated parameters is under progress, it is predicted that our code has a potential to calculate dynamic gain factors directly which is very important to predict whether cavitation instability occurs or not in advance.

2. Impeller-Diffuser Interaction in the Liquid Hydrogen Pump

The objective of this simulation is to propose a one-way coupled simulation method that combines CFD and structural analyses for impeller-diffuser interaction in a liquid hydrogen pump. The internal flow of the pump is computed to obtain the pressure fluctuations as shown in Fig. 5, and is fed to the structural analyses to compute the elastic wave propagation in the solid portion of the pump. In the internal flow computations, we assumed no feedback effect from the structural vibration of the pump to the internal flow.

The source fluctuations of the flow field are computed by a large-eddy simulation (LES) with the Dynamic Smagorinsky Model (DSM). Four cases of LES were carried out as shown in Table 1 for comparison with water tunnel experiments in the present simulations. Cases 1 through 3 are to check the influence of the phase difference between the blades of the first and second impeller. Case 4 is to check the influence of the impeller-diffuser distance. If the phase of the blades have a strong influence on the fluid-induced vibration, case 2 will show maximum pressure fluctuations. If the impeller-diffuser distance has a strong influence, we believe that case 4 will show minimum pressure fluctuations. All computations were carried out under non-cavitating conditions, to coincide with the experiments.

The main result in the present simulations is Fig. 6, where the velocity vector and static pressure distributions are shown. This figure shows that the origin of pressure fluctuations is evidently induced by impeller-diffuser interaction. Both instantaneous and time averaged results are presented in the figures. As shown in the figure, pressure fluctuations occur when the impeller blade interacts with the diffuser vane. We can also confirm that flow separation occurs at the pressure side of the full blades, as the primary flow of the impeller is on the right side of the splitter vanes. Thus, the jet-wake like velocity patterns at the impeller exit coincide with the number of the full blades.

We have confirmed that the phase difference between the first and second impellers, and the impeller-diffuser distance were the dominant phenomenon of the fluid-induced vibra-
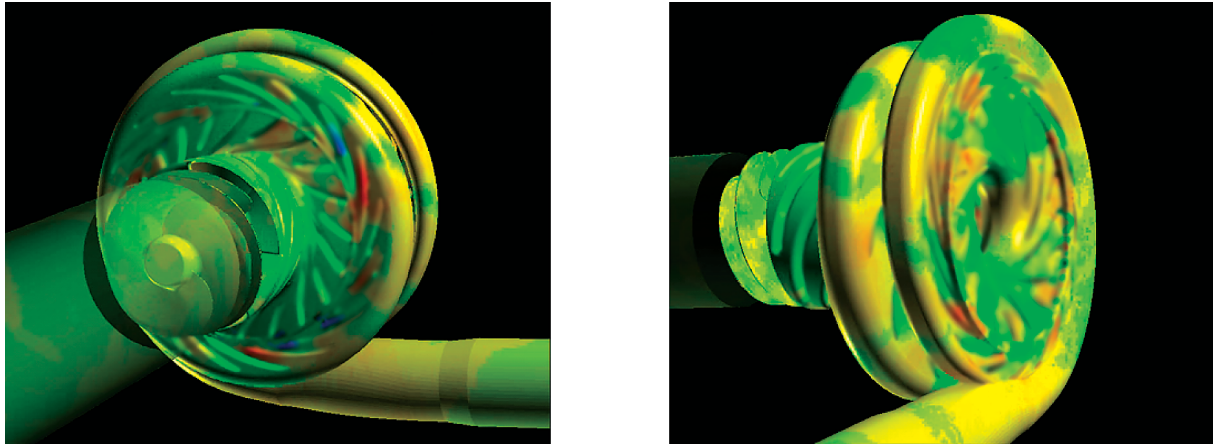

Fig. 5 Calculated pressure field in the liquid hydrogen pump. (left; front view, right; rear view)

Table 1 Computational conditions for comparison with water tunnel experiments.

Case No.	Inlet flow rate Q/Q_n	Phase difference between 1st and 2nd impeller blades	Impeller-diffuser distance (1st stage only)
1	1.00	7.4 degrees	Nominal
2	1.00	0.2 degrees	Nominal
3	1.00	14.6 degrees	Nominal
4	1.00	14.6 degrees	Expanded

Instantaneous velocity and pressure distribution   Time averaged velocity and pressure distribution

Impeller-diffuser interaction at the first stage

Instantaneous velocity and pressure distribution   Time averaged velocity and pressure distribution

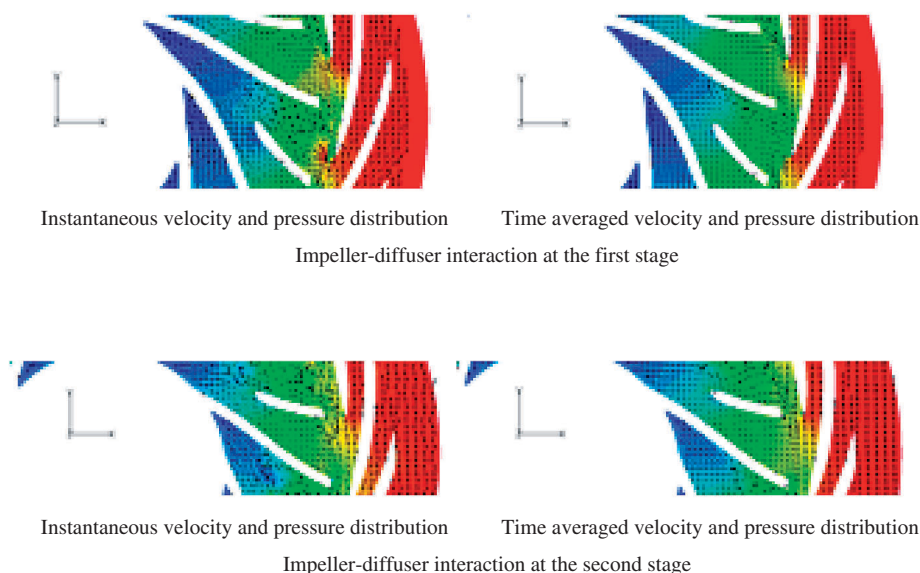Impeller-diffuser interaction at the second stage

Fig. 6 Impeller-diffuser interaction at the first stage (upper half) and second stage (bottom half).

tion. Although we spare the detailed comparison with experiments in this report, the computed pressure fluctuation and static pressure distribution agree fairly well with the measured data, which demonstrates that the proposed method can serve as a practical tool for predicting unsteady flows in a rocket engine pump, in the near future.

References

[1] C. Kato, and M. Ikegawa, "Large Eddy Simulation of Unsteady Turbulent Wake of a Circular Cylinder Using the Finite Element Method," ASME-FED, 117, 49–56 (1991).

[2] C. Kato, M. Kaiho, and A. Manabe, "An Overset Finite-Element Large-Eddy Simulation Method with Application to Turbomachinery and Aeroacoustics," Trans. ASME, 70, 32–43 (2003).

[3] K. Okita, Doctoral Thesis of Osaka University (in Japanese) (2002).

ロケットエンジン内部流れのシミュレーション

プロジェクト責任者

坪井　伸幸　　宇宙航空研究開発機構　情報・計算工学センター

著者

津田　伸一*1，山西　伸宏*1，加藤　千幸*2，清水　太郎*1

＊1　宇宙航空研究開発機構　情報・計算工学センター

＊2　東京大学　生産技術研究所

　国産ロケットの信頼性向上及び将来型宇宙輸送システムの開発に資するため、主要エンジン要素(燃焼器系・供給器系)で発生している諸問題を再現できるCFDコードを開発し、概念設計・システム評価・不具合対策等に使用し、試作・試験のサイクルを短くすることを目標にプロジェクトを進めている。本年度は供給器系に限定したうえで、特にターボポンプに起因する圧力振動や構造振動に着目した以下の2つの内容の解析を実施した。

　第一に、キャビテーション不安定現象に関する解析として、キャビテーションが生じる条件下において、入口に流量変動または圧力変動を与える計算を実施し、その際のキャビティ体積の応答を調べる解析を実施した。さらに、キャビテーション不安定の事前予測において重要となる動特性パラメータ(マスフローゲインファクタやキャビテーションコンプライアンス)の算出を試みた。結果として、キャビテーションが発達していない状態に限定されるという課題はあるものの、流量変動または圧力変動の計算からキャビティ体積の時間変化を検出可能であること、ひいては動特性パラメータの算出が可能であることを示した。これにより、実験的に求めることが極めて困難な動特性パラメータを数値実験的に求めることが可能となり、本計算手法がキャビテーション不安定現象の事前予測に利用できるポテンシャルを有していることを示した。

　第二に、ポンプインペラの動静翼干渉に起因した振動現象の解明を目的として、ポンプ全系内部流れを対象とした大規模LES解析を実施した。結果として、動翼と静翼の取り付け位相差に起因して生じる流れ場の違いが実験結果とも整合することを確認し、本計算手法が動静翼干渉にもとづく振動現象の解明にも大きく貢献できることを示した。

キーワード：ターボポンプ, LES, キャビテーション, 動的ゲイン, 軸振動, 動静翼干渉