Numerical Simulation of Rocket Engine Internal Flows

Group Representative
Hiroshi Miyajima Japan Aerospace Exploration Agency

Authors
Nobuhiro Yamanishi Japan Aerospace Exploration Agency
Taro Shimizu Japan Aerospace Exploration Agency
Nobuhide Kasagi Department of Mechanical Engineering, The University of Tokyo
Chisachi Kato Institute of Industrial Science, The University of Tokyo

Understanding the physics of the rocket engine internal flow is essential for developing a highly reliable space launch vehicle. The recent progress in computational fluid dynamics has changed our conventional approach for developing rocket engines. The LE-7A rocket engine, which is used in the booster stage of the Japanese H-2A rocket, has a fuel turbopump and an oxidizer turbopump to increase the pressure of the fuel and the oxidizer. This year, one of our main focuses was on the better understanding of the rocket engine turbopump. First, a large eddy simulation (LES) of a rocket turbopump inducer in non-cavitating and cavitating flows were carried out. Results have shown that the backflow region rotates with the inducer, at a 13% rotational speed of the inducer, which is consistent with experimental results that have shown rotational speeds of 10 to 20% of the inducer. We have also carried out LES analysis of a centrifugal impeller, another main part of the turbopump. In this study, emphasis was placed on the pressure fluctuations that result from the interaction between the rotating impeller blades and stationary diffuser vanes. A gas turbine drives the above-mentioned inducer and impellers and composes one of the major components of the rocket engine turbopump. In order to clarify the transition and/or separation mechanism that has not yet been fully understood, we also performed large eddy simulation of transitional boundary layers in a model turbine cascade. Finally, a turbulent channel flow at $Re_\tau = 1160$ is studied by means of direct numerical simulation (DNS), aiming at improvement of turbulence models to be applicable in high Reynolds number wall-turbulence.

Keywords: H-2A rocket, LE-7A rocket engine, rocket turbopump, large eddy simulation, direct numerical simulation

Report of your result

Understanding the physics of the internal flow of a rocket engine is essential for developing a highly reliable space launch vehicle. Until recently, the development of Japanese rockets were largely based on trial & error, i.e.: an iterative cycle of the trial design and experimental verification. The recent progress in computational fluid dynamics has changed this approach, as numerical simulation is playing a major role for the development of rockets and rocket engines built today. The H-2A launch vehicle, Japan's present expendable launch vehicle, which is capable of carrying a 4-ton-class payload into a geostationary transfer orbit, has been under development since 1995. The LE-7A engine, which is used in the booster stage of the H-2A rocket, provides a thrust of 1100kN using liquid hydrogen and liquid oxygen as propellants. The LE-7A fuel turbopump (FTP) and oxygen turbopump (OTP) were also developed to decrease manufacturing costs, as well as to achieve higher reliability utilizing experience gained in the development of turbopumps in the past. These turbopumps have a single stage turbine, a two stage centrifugal impeller and a single inducer, which is required for high durability and stable operation under all operating conditions during flight.

In order to increase the reliability of the LE-7A engine of the H-2A launch vehicle and apply its knowledge to future space vehicles, we have conducted numerical simulations of liquid rocket engine internal flows. This year, one of our main focuses was on the better understanding of the rocket engine turbopump.

First, a large eddy simulation (LES) of a rocket turbopump inducer in non-cavitating and cavitating flows were carried out. The computation takes full account of the interaction between the rotating inducer and the stationary casing by using a multi-frame-of-reference dynamic overset grid approach[1]. A streamline-upwind finite element formulation with second-order accuracy both in time and space is used to discretize the governing equation. It is implemented in parallel by a domain-decomposition-programming model. The evolution of cavitation is represented by the source/sink of vapor phase in the incompressible liquid flow. The pressure-velocity coupling is based on the fractional-step method for incompressible fluid flows, in which the compressibility is taken into account through the low Mach number assumption. The basic design of this inducer is similar to the one used in the LE-7 and LE-7A rocket engine liquid oxygen turbopump. Particular emphasis was placed on the large vortical structures that appear at off-design conditions (flow coefficient $\phi = 0.05$).

Figure 1 is the instataneous meridional velocity (C_m) distribution at $z/D_t = -1.85$, -0.95, and -0.05 for $\phi = 0.05$. Shown in the figure are images of four continous revolutions. We can confirm from the figure that the backflow region rotates with the inducer, at a 13% rotational speed of the inducer. This is consistent with experimental results that have shown rotational speeds of 10 to 20% of the inducer.

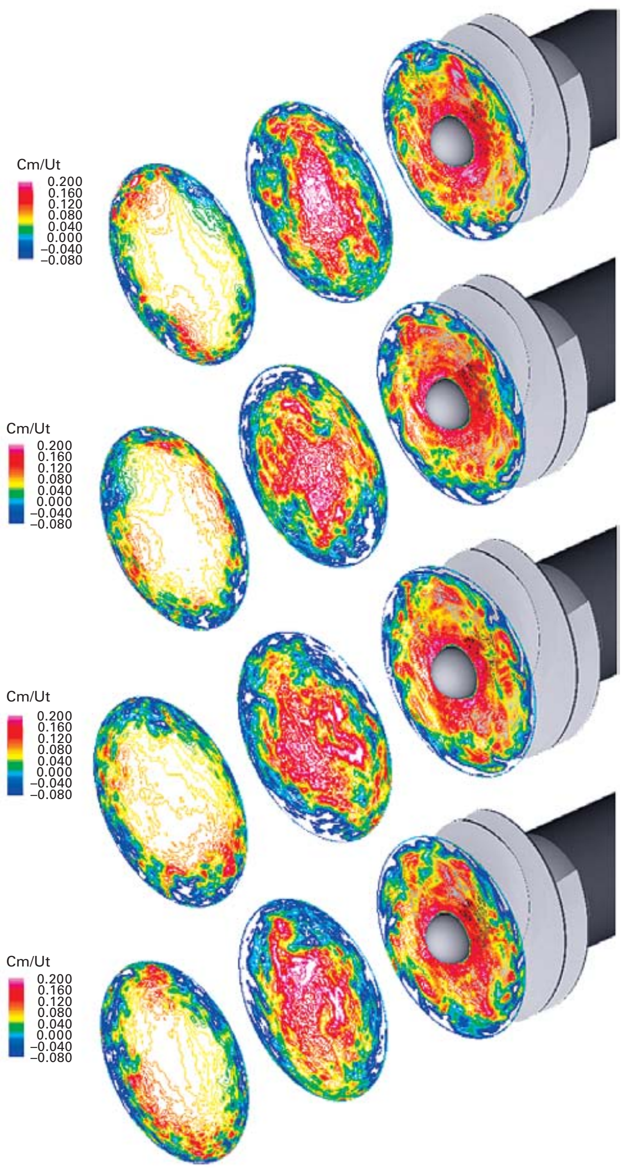

Fig. 1 Instantaneous meridional velocity distribution at $z/D_t = -1.85$, -0.95, and -0.05 for four continous revolutions (flow coefficient $\phi = 0.05$). Blue colored region indicates the inlet backflow. D_t and U_t are inducer tip diameter and velocity, respectively.

Further information on this research can be found in ref. 2.

As already mentioned, centrifugal impellers are the main part of a turbopump for the present rocket engines. Understanding of the flow physics that is taking place in the impellers is also a key for achieving higher reliability of the engine. We therefore performed large eddy simulation of flow in a centrifugal impeller at two different flow-rate ratios. In this study, emphasis is on the pressure fluctuations that result from the interaction between the rotating impeller blades and stationary diffuser vanes. Figure 2 shows typical instantaneous flow field in the impeller and diffuser where instantaneous static pressure is indicated by its color. Intense fluctuations of the static pressure are seen only in the diffuser vanes at the design flow-rate ratio (top left) while they are extended to far upstream of the impeller blades at off-design flow-rate ratio (top right). The frequency spectra of these pressure fluctuations show an excellent agreement with the measured equivalent as shown in the bottom figure.

A gas turbine drives the above-mentioned inducer and impellers and composes one of the major components of the rocket engine. The aerodynamic performance of such a turbine is greatly affected by the transition and/or (possible) separation of the boundary layer that develops on the blade's surface of the turbine. In order to clarify the transition and/or separation mechanism that has not yet been fully understood, we also performed large eddy simulation of transitional boundary layers in a model turbine cascade. In this simulation, effects of the incident turbulence are also taken into account as in the case of the actual operation conditions.

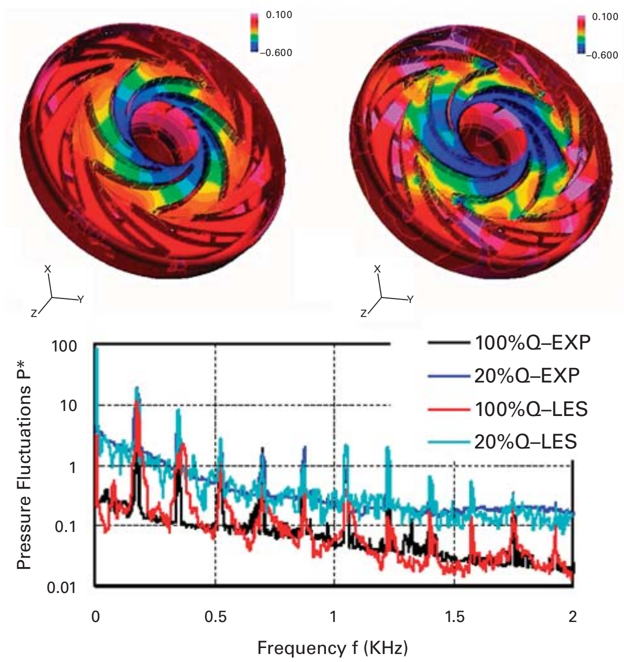

Fig. 2 Instantaneous distributions of static pressure in a centrifugal impeller at two different flow rate ratios (top left: design point, top right: off-design point) and power spectra of the pressure fluctuations (bottom).

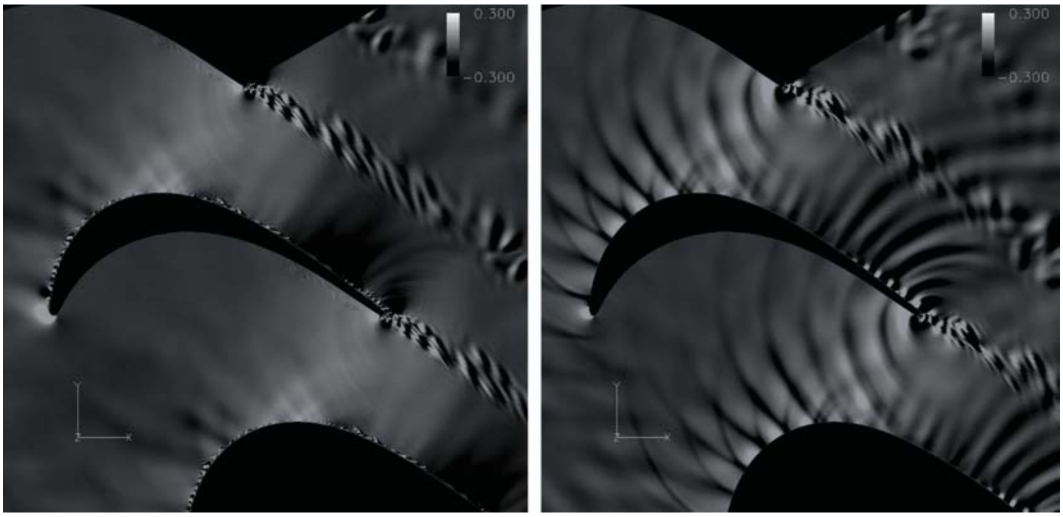

Fig. 3 Instantaneous distributions of static pressure in a turbine-blade cascade with (left) and without (right) incident free-stream turbulence

Figure 3 presents one snap-shot of the cascade flows. When the incident turbulence is considered (left figure), the transition process of the boundary layer is essentially different from natural transition (right figure). Although not shown here, the computed pressure distribution around the blade quantitatively agrees with the measurements. Investigation on the detailed mechanism of the transition is currently underway.

Finally, some fundamental characteristics of a turbulent channel flow at $Re_\tau = 1160$ are studied by means of direct numerical simulation (DNS). Our aim is to accumulate the essential knowledge on high-Reynolds number wall-turbulence, which can be used for improvement of turbulence models, such as subgrid-scale model in above-mentioned LES.

Up to now, various Reynolds number effects in wall-turbulence have been studied. It is well known that near-wall vortices play an important role in the transport mechanisms in wall-turbulence, at least, at low Reynolds number flows[3]. At higher Reynolds numbers, however, characteristics of near-wall coherent structures remain unclear. Adrian *et al*.[4] show that packets of large-scale hairpin vortices are often observed in high-Reynolds-number wall-turbulence. In the present study, the dynamics of the near-wall coherent structures and large-scale structures are evaluated through direct numerical simulation (DNS) of turbulent channel flow at moderate Reynolds numbers.

The numerical method used in the present study is almost the same as that of Kim *et al*.[5]: a pseudo-spectral method with Fourier series is employed in the streamwise (x) and spanwise (z) directions, while a Chebyshev polynomial expansion is used in the wall-normal (y) direction. The friction Reynolds number Re_τ is 1160, which is the world record concerning the DNS of turbulent channel flow. Hereafter, u, v, and w denote the velocity components in the x-, y-, and z-directions, respectively. Superscript ($^+$) represents quantities non-dimensionalized with the wall friction velocity u_τ and the kinematic viscosity ν.

Figure 4 shows a ($x - z$) plane view of an instantaneous partial flow field at $Re_\tau = 1160$, in which the vortices identified with isosurfaces of the second invariant of the deformation tensor ($Q^+ = -0.02$) are visualized. It is found that the vortices form clusters in low-speed regions, and that some hairpin vortices are observed in high-speed regions.

Figure 5 shows a ($y - z$) cross-stream plane of an instantaneous partial flow field, in which contours of the streamwise velocity fluctuation u' and vortices ($Q^+ < -0.005$) are visualized, in order to examine the relationship between the near-wall vortices and the large-scale outer-layer structures. The near-wall vortices are located between low- and high-speed streaky structures as same as those in low Reynolds number flows[6]. Away from the wall, low/high-speed large-scale structures appear, and the vortices are clustered preferably in the low-speed regions. The streaky structures, of which spanwise spacing is about $100\ \nu / u_\tau$, exist only near the wall ($y^+ < 30$), while the large-scale structures exist from the center of the channel to the near-wall region ($y^+ \sim 30$). Note that the contribution of the large-scale structures to the Reynolds stress is much larger than that of the clustering vortices in the outer layer.

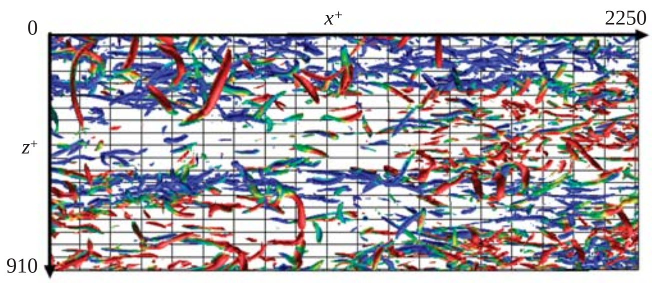

Fig. 4 Plane view of vortices at $Re_\tau = 1160$. Iso-surface, $Q^+ = -0.02$; blue to red, $u'^+ = -1$ to $u'^+ = 1$.

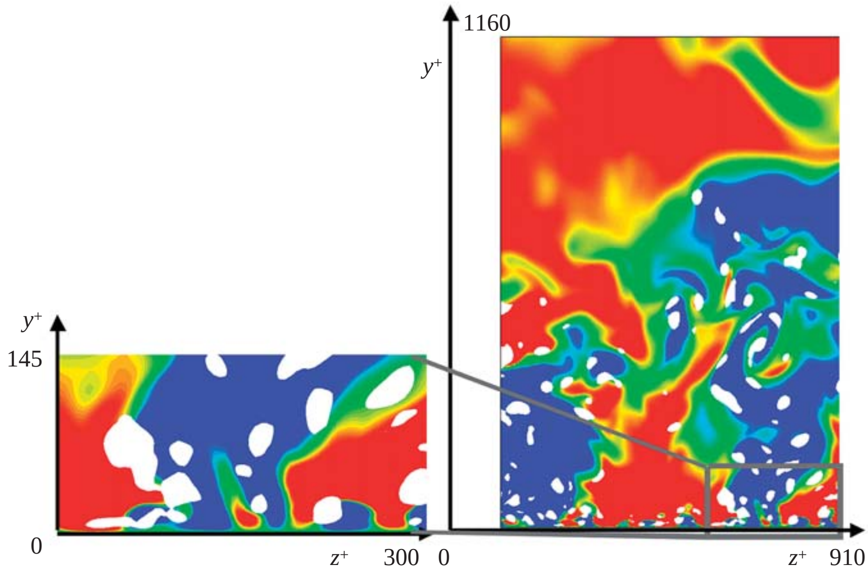

Fig. 5 Cross view of instantaneous velocity field at $Re_\tau = 1160$. Contours of the streamwise velocity fluctuation, blue to red, $u'^+ = -1$ to $u'^+ = 1$; white, $Q^+ < -0.005$.

References

[1] Kato, C., Shimizu, H., and Okamura, T., "Large Eddy Simulation of Unsteady Flow in a Mixed-Flow Pump," ASME FEDSM 99-7802.

[2] Yamanishi, N., Kato, C., and Matsumoto, Y., "LES Analysis of a Rocket Turbopump Inducer in Non-cavitating and Cavitating Flows," ASME FEDSM 2003-45406.

[3] Robinson, S. K., "Coherent motions in the turbulent boundary layer," Annu. Rev. Fluid Mech., 23, 601 (1991).

[4] Adrian, R. J., Meinhart, C. D., and Tomkins,C. D., "Vortex organization in the outer region of the turbulent boundary layer," J. Fluid Mech., 422, 1 (2000).

[5] Kim, J., Moin, P., and Moser, R., "Turbulence statistics in fully developed channel flow at low Reynolds number," J. Fluid Mech., 177, 133 (1987).

[6] Kasagi, N., Sumitani, Y., Suzuki, Y., and Iida, O., "Kinematics of the quasi-coherent vortical structure in near-wall turbulence," Int. J. Heat Fluid Flow, 16, 2 (1995).