Cost-effective CFD for vortex shedding from a trailing edge of hydraulic turbine blade

Sadao Kurosawa, Masahiko Nakazono
Hydraulic Machinery Group, Rotating Machine Technology R&D Department
Power and Industrial Systems R&D Center, Research & Development Div.
Toshiba Corporation
20-1, Kansei-cho, Tsurumi-ku, Yokohama 230-0034, Japan

sadao.kurosawa@toshiba.co.jp, masahiko.nakazono@toshiba.co.jp

Abstract. This paper describes the method for analysing the vortex from a trailing edge of the blade with high accuracy using CFD. As for the analysis method, RANS and LES hybrid model, which is a combination of RANS with a low calculation load and LES with a high calculation load, was used on understanding the turbine-scale phenomenon with a relatively low calculation load. As for the first step its application as a turbine runner, we conducted the case study in which the blade model was fixed in a uniform and we confirmed that the present CFD could be predicted the vortex shedding phenomena, such as power spectra of velocity fluctuations, wake fluctuation, vortex shedding pattern and hydraulic force acting on the blade etc.

1. Introduction
Hydraulic turbine manufacture were faced with the problem of turbine runner vibration, the excitation force being produced by the vortices which regularly shed from the trailing edge of the blade and which arrange themselves in the von Karman vortex street downstream. The situation becomes serious when the excitation force frequency coincides with one of the resonant frequency of the runner. As for this problem, a lot of research were conducted and it was clear that the detrimental resonant vibration in turbines could be eliminates by a modification of the trailing edge geometry. [1] [2] [3]

However the trailing edge geometry for countermeasure of the von Karman vortex is not the best for the turbine efficiency. It is desired that the trailing edge geometry is optimized considering the resonant vibration of the von Karman vortex and the turbine efficiency.

As for CFD of the Karman vortex from a trailing edge of the blade, it needs to apply Large Eddy Simulation (LES) since the unsteadiness of the flow field is strong and the energy cascade phenomenon is dominant. [4] However LES needs huge grid number, RANS (Reynolds-Averaged Navier-Stokes) and LES hybrid model is practical. Yamada et al investigated the Karman vortex from a trailing edge of the blade by using some RANS-LES hybrid models. [5] It was pointed that Detached Eddy Simulation (DES) could not catch the vortex shedding phenomena since the region near the trailing edge was judged as RANS region. It could be predicted to assign the LES region properly by using Embedded LES (ELES). However there are some problems for practical use. It needs DES calculation to get the information for the judgment RANS and LES region, namely computational cost is double. The parameter for the judgement is experiential one and is not general.
In this situation, as for the first step its application as a turbine runner, we conducted the case study in which the blade model was fixed in a uniform. As for CFD, on the basis of ELES which RANS and LES region decomposition is simple, it was carried out to use three kind of LES, in which first one was Wall Adapting Local Eddy Viscosity Model (WALE), second one was Dynamic Smagorinsky Model (DSM) and third one was Wall Modeled Large Eddy Simulation (WMLES). Moreover Stress Blended Eddy Simulation (SBES), which is reported that the judgment RANS and LES region is improved, was conducted. And then, the prediction accuracy for the vortex shedding phenomena from a trailing edge of the blade was evaluated to compare with the experiment data.

2. Target for Blade and Experiment
The experiment data for the blade was measured by Bourgoyne et al. using the two-dimensional cross sectional model at the Large Cavitation Channel (LCC). The test section geometry and the blade section are shown in Figure 1, 2. [6][7] It shows the velocity power spectra (vertical Blade flow velocity fluctuations) measured at 1%, 7% blade chord length downstream from the blade trailing edge. And also it was measured the static pressure on the blade surface, the flow near the trailing edge (boundary layer, wake), the instantaneous swirling strength field by PIV, etc. The measurement results for Reynolds number of $4 \times 10^6$ were used for CFD validation. Figure 2 also defines the coordinate frames used in this paper. The streamwise coordinate, $x$, is defined by the tunnel axis, and vertical coordinate, $y$, is taken normal to $x$. The vertex of the leading edge is defined as the coordinate origin.

![Figure 1. The test section geometry. [6]](image1)

![Figure 2. The blade section.[6]](image2)
3. Analysis Method for Vortex Shedding Phenomena

Two kinds of RANS and LES hybrid models were used to resolve the vortex shedding phenomena from the blade trailing edge. One of them is ELES and another one is SBES. The CFD code is commercial finite volume code “ANSYS Fluent” version17.2. [8]

As for ELES, the RANS region where the boundary layer can be resolved with high accuracy was assigned upstream of the blade surface and the LES region was assigned downstream from the trailing edge where vortices are generated. Turbulence model for the RANS region is SST k-ω model. Sub Grid Scale (SGS) model for the LES region are WALE, DSM and WMLES.

In WALE, the turbulence viscosity is calculated by the following equation. It is considered the damping effect near the wall and the fault of Smagorinsky LES model is overcome.

\[ \nu_t = \left( C_s \Delta \right)^2 \left( \frac{S_{ij} S_{ij}}{S} \right)^3 \left( \frac{S_{ij} S_{ij}}{S} \right)^{-1/4} \]

Where, \( C_s \) is the Smagorinsky coefficient, \( \Delta \) is the size of the grid filter and \( S_{ij} \) is the strain–rate tensor. In WALE calculation, \( C_s \) is constant value (\( C_s = 0.325 \)).

In DSM, the turbulence viscosity is calculated by the following equation. The Smagorinsky coefficient is determined locally in time and space. The fault of Smagorinsky LES model, such as transitional flow, shear flow etc., is overcome.

\[ \nu_t = \frac{1}{2 \Delta^2} \left( \frac{M_{ij}}{M_{ii}} \right) \]

\[ M_{ij} = \frac{\partial \psi}{\partial u_i} \frac{\partial \psi}{\partial u_j} - \frac{\partial \psi}{\partial u_j} \frac{\partial \psi}{\partial u_i} + \frac{1}{3} \frac{\partial \psi}{\partial u_k} \frac{\partial \psi}{\partial u_k} \]

\[ \nu_t = f_D \min \left( (ky)^2, (C_{SMGL} \Delta)^2 \right) S \]

Where \( y \) is the wall distance, \( \kappa \) is the von Karman constant and \( f_D \) is a near-wall damping function. Near the wall, the min-function selects the Prandtl mixing length model whereas away from the wall it switches over to the Smagorinsky model.

As for SBES, the shielding function \( f_S \) is used to explicitly switch between different turbulence model formations in RANS and LES mode. The definition of the shield function has not been disclosed. However it has strong shielding and it is possible to maintain a zero pressure gradient RSNS boundary layer on any grid. ANSYS recommend relative to other RANS and LES hybrid models. The turbulence viscosity is calculated by the following equation.

\[ \nu_t^{SBES} = f_S \cdot \nu_t^{RANS} \cdot (1 - f_S) \cdot \nu_t^{LES} \]

Turbulence model for the RANS region is used SST k-ω model. Sub Grid Scale (SGS) model for the LES region is used DSM.
As for the discretizing method, Cell based finite volume method was used and the convective term of momentum equation was approximated by second order upwind for RANS and bounded central difference (BCD) for LES. Transient term of governing equation was approximated by second order. And also, as for numerical algorithm to solve the algebraic finite volume equations, the pressure implicit split operator (PISO) method was used. Moreover, the time increment is set so that courant number is under one and the computation was started from an initial field calculated by steady RANS. Total computation is divided into two parts: the first one is to reduce the initial disturbance and the second one is to do the time-averaged processes and the evaluation of vortex shedding phenomena, such as power spectra of velocity fluctuations, wake fluctuation, vortex shedding pattern.

4. Computational Grid and Boundary Condition

Overview of the computational grid is shown in Figure 3. Zoom up figure near the trailing edge is shown in Figure 4. The computational domain is defined between the blade chord length (C) upstream from the leading edge and the blade chord length downstream from the trailing edge. The height size of the computational domain is 1.4C and the span size is 0.1C. The hexahedral elements are used to have as high resolution as possible. The grid density around the blade surface is high to use O type topology and the none-dimensional minimum cell wall distance become \( y^+ < 1 \). The grid density near the trailing edge between 0.1C upstream and 0.2C downstream is especially high to resolve the vortex shedding phenomena. The grid size in this region is determined on the basis of the turbulence eddy size estimated by RANS calculation and the grid size is about half size of taylor microscale. Total number of grids is 8162584. As for ELES, the boundary of RANS region and LES region is set 0.1C upstream from the trailing edge.

The computational boundary conditions were applied at the inlet surface and at the outlet surface of the computational domain. About the inlet boundary condition, it was assumed the uniform velocity distribution. As for the outlet boundary condition, the average pressure was set to fix. Furthermore, about the surface of blade wall, the non-slip boundary conditions were prescribed, i.e. the velocity components were set to zero. The upper and lower surfaces of the computational domain were set as the free slip wall boundary. The side face boundary was set as the periodic boundary.

As for the interface boundary of ELES_RANS region and ELES_LES region, the perturbations on the inlet boundary of LES region is generated by vortex method. [9] In the vortex method, the vortices are assigned on the inlet boundary of LES region and the vorticity of the vortices are calculated by \( \kappa \) and \( \varepsilon \) on the outlet boundary of RANS region. And the perturbation velocity is calculated by Biot-Savart law.
5. Simulation Results and Discussion

5.1. Time averaged flow field

Time averaged pressure fields are shown in Figure 5. The static pressure coefficient \(C_p\) on the blade surface is shown in Figure 6. \(C_p\) is defined as
\[
C_p = \frac{2(P - P_0)}{\rho U_0^2}
\]
where \(P\) is the static pressure, \(\rho\) is the water density and \(P_0\) is the reference pressure measured at the same upstream location as the reference velocity \(U_0\). Furthermore, to validate the CFD results, the experiment data is also plotted in Figure 6.

From Figure 5, it can be seen that all results are similar and static pressure distribution is depend on the blade camber shape. Namely, the upper surface of the blade is the suction side and the lower surface is the pressure side. From Figure 6, it can be recognized that the simulation results are little higher compared with the experiment. This is why the side wall of the computational grids is applied the periodic condition and the friction loss is not generated. Considering the side wall effect, the simulation results are reasonable. As for the pressure gradient on the suction surface near the trailing edge that is affected the vortex shedding phenomena, ELES_WALE and ELES_DSM are little more adverse compared with other results.
Figure 6. The static pressure coefficient ($C_p$) on the blade surface.

Time averaged velocity fields are shown in Figure 7. The boundary layer flow at $x/C=0.93$ and the wake flow near the trailing edge are shown in Figure 8, 9. Furthermore, to validate the CFD results, the experiment data is also plotted in Figure 8, 9.

From Figure 7, it can be seen that all results are similar and static velocity distribution is depend on the blade camber shape as well as the time averaged pressure field. As for the boundary layer flow at $x/C=0.93$, it can be found that all results are similar and the boundary layer thickness on the suction surface is predicted somewhat more thickly. On the other hand, the boundary layer thickness on the pressure surface is predicted somewhat thinness. However it is thought that the simulation results are reasonable. The counters of the SBES shield function near the trailing edge is shown in Figure 10. In this figure, the blue region is the RANS region and the red region is the LES region. From this figure, it can be understood that the boundary layer flow at $x/C=0.93$ is RANS region and that is why the SBES results are almost same as the ELES results. As for the near wake flow at $x/C=1.01$, it can be found that the width of the wake is predicted somewhat more thickly and the position of the minimum velocity is good agreement with the experiment. About the prediction error for the wake width, it is related to the prediction error for the boundary layer flow near the trailing edge.
Figure 8. The boundary layer flow at x/C=0.93.
(Streamwise velocity normalized by main flow velocity U_e)

Figure 9. The wake flow near the trailing edge. (x/C=1.01)
(Note)
\[ \Delta U = U - U_e^{ss} \]
\[ U_e^{ss} = \text{maximum velocity of main flow at suction side} \]
\[ = \beta \times U_0 \]
\[ \beta = 0.896 : \text{referred from paper [1]} \]

Figure 10. The SBES shield function near the trailing edge.

Therefore it is thought that ELES and SBES can predict the time averaged flow field that is based on the vortex shedding phenomena.
5.2. Vortex shedding phenomena

Instantaneous vorticity distributions near the trailing edge are shown in Figure 11. In this figure, to validate the CFD results, the vorticity in the PIV measurement is also shown. And also clockwise rotating fluid appears white in the PIV measurement and blue in the CFD results. Counterclockwise rotating fluid appears black in the PIV measurement and red in the CFD results. From this figure, it can be seen that the vortex size of SBES is overestimated and the vortex size of the PIV measurement is similar to the ELES results. As for the ELES results, it can be seen that the vortex size became large in order ELES_WALE, ELES_DSM, ELES_WMLES. About this matter, it is thought that it is depend on the turbulence viscosity shown in Figure 12. Namely the vortex size is in proportional to the turbulence viscosity.

For the time history data of the vertical flow velocity fluctuation at x/C=1.01,1.07, a Fourier transform was conducted. A comparison of the respective CFD calculation results of the power spectra and the experiment data is shown in Figure 13. From this figure; it can be found that SBES can predict the peak frequency of the likely Karman vortex. However SBES overestimate the level of the power spectra at x/C=1.07. As for this matter, it is consistent in the result that SBES overestimate the vortex size. On the other hand, as for ELES, ELES_WALE overestimate the peak frequency and underestimate the level of the power spectra. In order ELES_WALE, ELES_DSM, ELES_WMLES, the CFD results become closer to the experiment. It can be recognized that ELES_WMLES predict the vortex shedding phenomena with high accuracy. As for this matter, it is thought that the ELES_WMLES results whose turbulence viscosity is larger than other, is closer to the experiment because the process for vortex mixing and union is promoted by increasing the turbulence viscosity.
Figure 12. Instantaneous turbulence viscosity distribution near the trailing edge (unit: Pa/s).

Figure 13. Power spectra of vertical velocity fluctuation.
6. Conclusions
Some RANS and LES hydride models were applied for the vortex shedding phenomena on the blade and the prediction accuracy was evaluated. The obtained results are as follow:

(1) As for the time averaged flow field that is based on the vortex shedding phenomena, the prediction results for ELES and SBES are appropriate. Therefore it is thought that it is cost-effective to apply RANS for the region around the blade.
(2) As for the vortex size, the result of SBES is overestimated and the results of ELES are reasonable.
(3) As for the peak and level of the power spectra for the vortex shedding, SBES can predict the peak frequency with high accuracy however SBES overestimate the level of the power spectra at the downstream region.
(4) As for ELES, ELES_WALE overestimate the peak frequency and underestimate the level of the power spectra but in order ELES_WALE, ELES_DSM, ELES_WMLES, the CFD results become closer to the experiment. And the result of ELES_WMLES is the best for the prediction accuracy for the vortex shedding phenomena.

From now on, we will develop high performance and high reliability hydraulic turbine by applying this method to runner design.

Acknowledgment
This CFD project was carried out by the help of project members in Power and Industrial Systems R&D Center of Toshiba Corporation and the support of members in CAE Technology Center of Toshiba Information System Corporation. The authors would like to acknowledge Kiyoshi Matsumoto, Toru Kuriyama, Yoshihiro Ishikawa for helpful discussion and for suggesting the problem and Naofumi Shibata, Yoshiaki Hayashi for supporting the CFD work.

References
[1] Gunnar Heskestad, et al., “Influence of Trailing-Edge Eeometry on Hydraulic-Turbine-Blade Vibration Resulting From Vortex Excitation”, Journal of Engineering for Power, ASME, April 1960
[2] Liess, C., “Flow induced stress fluctuations in stay vanes of large hydraulic turbines”, International Conference on Flow Induced Vibrations, England
[3] R.M. Donaldsen, “Hydraulic-Turbine Runner Vibration”, Trans. ASME, vol. 78, 1956
[4] Seung-Jae LEE, et al., “Numerical Investigation on Vortex Shedding from A Hydrofoil with a Beveled Trailing Edge”, Modeling and Simulation in Engineering, Volume 2015, Article ID565417
[5] Takuyoshi Yamada, et al., “Study on the Calculation Method for Unsteady Vortex from Trailing Edge of Blade”, AICFM13, Vol. 13, 2015
[6] Bourgoyne, D.A., et al., “Time averaged flow over a hydrofoil at high Reynolds number”, Journal of Fluid Mechanics, Vol. 496, (2003), pp. 365-404
[7] Bourgoyne, D.A., et al., “Vortex shedding from a hydrofoil at high Reynolds number”, Journal of Fluid Mechanics, Vol. 531, (2005), pp. 293-324
[8] F.R. Menter, “Best Practice: Scale-Resolving Simulations in ANSYS CFD”, ANSYS Germany GmbH, November 2015
[9] Mathey, F., et al., “Specification of LES Inlet Boundary Condition Using Vortex Method”, 4th International Symposium on Turbulence, Heat and Mass Transfer, Antalya Turkey, 2003.