Fully-resolved large eddy simulation of transitional flow in inter-stage bush of a centrifugal pump

R Prunieres¹, C Kato², Y Yamade³ and T Nagahara¹

¹ Industrial Products Business Unit, Hitachi Ltd., Japan
² Institute of Industrial Science, The University of Tokyo, Japan
³ Mizuho Information & Research Institute Inc., Japan

Email: romain.prunieres.gx@hitachi.com

Abstract. When predicting the performances of a centrifugal pump by eddy-resolving computational fluid dynamics (CFD) methods such as Large-Eddy Simulation (LES), it is important that the phenomenon of interest and associated flow eddies are accurately computed. Despite recent performance improvement of high-end computers, it is still a challenge to compute a whole multi-stage pump with leakage parts by applying fully-resolved LES and to directly compute active eddies in the turbulent boundary layers, due to grid size requirements. As a part of the analysis of the flows in a multi-stage centrifugal pump by LES, present research focuses on the flow in the pump inter-stage bush. It aims at determining the flow regime, the associated eddies in the bush, and a better understanding of the flow in a grooved inter-stage bush to allow future design optimisations. To assess the existence of the turbulent boundary layer for different Reynolds numbers, focus was placed on the appearance of streamwise vortices which are the smallest active eddies in the turbulent boundary layer. It was observed that these vortices start to develop from \( Re_\omega = 1800 \) although the velocity profile looks of a laminar boundary layer. Finally, based on CFD results, the effectiveness of the stage bush in terms of minimizing the leakage flow is investigated for the different Reynolds numbers. It is found that the current design shows the maximum effectiveness for \( Re_\omega = 900 \), and its effectiveness decreases at a higher Reynolds number.

1. Introduction

The control of the performance-curve slope is very important for pump designers because the part load instabilities, characterised by a local dent (head drop with decreasing flow rate), can result in abnormal operating conditions, which may damage both the pump and the system. However, although numerous studies have been performed, both experimentally and numerically, the onset and the mechanism of such instability is still not well understood.

The present work is a part of research that aims at applying fully-resolved Large Eddy Simulation (LES) to analyse performance-curve stability of a centrifugal pump. Fully-resolved LES computes directly the turbulent boundary layer and its smallest active eddies, which are the streamwise vortices in the inner layer of the boundary layer. By using fully-resolved LES, the authors intend to consider all phenomenon involved in the performance-curve head drop and hence to have a better understanding of its mechanism.

Additionally, it is also important to include, in the computation domain, all the pump components that could possibly influence the performance-curve stability behaviour. To this end, leakages parts
(wear ring upstream the impeller and inter-stage bush downstream the impeller) should also be included in the computed domain. However, fully resolving the turbulent boundary layer in these parts can easily lead to increase the total number of elements to solve the flow in the pump. Despite recent increase of computer performances and the availability of supercomputers, it is still a challenge to compute the flow when the number of the elements exceed ten billion. Therefore, the usefulness of using fully-resolved LES in these parts should also be evaluated.

Hence, the present paper focuses on the investigation of the flow in the inter-stage bush, equipped with radial grooves, of a multi-stage centrifugal pump. It aims at analysing the flow in the grooved inter-stage bush, determining when the flow becomes turbulent and consequently when it is needed to apply fully-resolve LES to this part of the pump. Four Reynolds numbers, based on the circumferential velocity of the pump shaft and the radial clearance between shaft and casing, have been considered here: \(Re_e=450, 900, 1800\) and \(3600\), to cover the possible flow conditions of the whole-pump computations in the future. As the flow in a grooved inter-stage bush is typically transitional flow, and because Reynolds Averaged Navier-Stokes (RANS) simulation accuracy is mostly degraded when transitional flow is of interest, LES has been used in this study.

2. Numerical Setup

2.1. Governing Equations

The governing equations considered here are the spatially filtered incompressible continuity (equation 1) and Navier-Stokes (equation 2) equations:

\[
\frac{\partial \tilde{v}_i}{\partial x_i} = 0 \tag{1}
\]

\[
\frac{\partial \tilde{v}_i}{\partial t} + \frac{\partial \tilde{v}_i \tilde{v}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \tilde{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial \tilde{v}_i}{\partial x_j} + \frac{\partial \tilde{v}_j}{\partial x_i} \right) - \tau_{ij} \right] \tag{2}
\]

Where \(\tilde{v}\) is the filtered velocity, \(\tilde{p}\) the filtered pressure, \(\rho\) and \(\nu\) are the fluid density and kinematic viscosity, respectively, and \(\tau_{ij}\) is the sub-grid scale (SGS) stress tensor defined as:

\[
\tau_{ij} = \left( \tilde{v}_i \tilde{v}_j - \tilde{v} \tilde{v} \right) \tag{3}
\]

The last term in the right-hand side of equation 3 expresses the effect of the small-scale eddy motion and has to be modelled. In the present research, the effect of the small scales is mimicked using a dynamic Smagorinsky model [1][2][3].

2.2. Software

Computations are done using the open-source software FrontFlow/Blue (FFB) developed by The University of Tokyo [4][5]. FFB is based on finite element method and allows massive parallelisation of the computations. This software has already been used successfully to analyse pump internal flows [6] or rotating stalls [7] among other applications.

To overcome the potential difficulties to handle large files generated by a LES computation and to take advantage of FFB parallelisation capability, post-treatment was done using the open-source software VisIt developed by the Lawrence Livermore National Laboratory [8]. VisIt offers the possibility to remotely access computation results files and allows parallel visualisation for large-scale data spread among several CPUs.

2.3. Computing Resources

All the computations for this research were performed on K computer at the RIKEN Center for Computational Science (R-CCS) located in Kobe in Japan. K computer is composed of 82,944 computing nodes with 8 cores and has a peak performance of 10.6 PFLOPS in total and 128 GFLOPS per node.
2.4. *Computational Model, Mesh and Settings*

The inter-stage bush analysed in this paper is equipped for a multi-stage centrifugal pump with a universal specific speed $\omega_s = 0.65$, for which performances were previously analysed by using RANS simulation [9].

Figure 1 presents a schematic of the computational domain. The bush is composed of several radial grooves to achieve high pressure loss and then limit leakage flow rate of the pump. The space between each groove is known as “liner”. Downstream the bush, the model also includes the impeller back chamber.

In this study, a coarse mesh composed approximately of 70 million hexahedral elements was used (Figure 2) to perform a preliminary analysis regarding the capability of the mesh to capture streamwise vortices if any. Using this mesh, one or two refinements (denoted, respectively, by 1R and 2R) were done to resolve the turbulent boundary layer that develops on the shaft and the stationary walls of the bush. Because mesh refinement is performed in FFB by dividing sides of each elements into two for the 3 directions, the total number of elements is approximately 4.5 billion after two refinements.

At the bush inlet, time-averaged axial and tangential velocity components obtained by preliminary LES for the whole pump configuration were set while the pressure was set to zero at the outlet. The shaft was modelled as a rotating wall of constant angular velocity while the casing was set as a stationary wall. Both walls were set with a non-slip condition. To change the Reynolds number, fluid kinematic viscosity was modified. One shaft revolution was divided into 3,600 time steps for the non-refined mesh. For computations with refined mesh, the number of time steps for one shaft revolution was multiplied accordingly to keep an appropriate CFL number. These computational settings are summarised in Table 1 along with CPU time per time step. For computation with twice-refined mesh (2R), approximately 10 hours were needed for one revolution by using 8192 computing nodes.

| Number of Mesh Refinements | Computing Nodes | Time Steps Per Shaft Revolution | CPU Time/Step [s] |
|----------------------------|-----------------|---------------------------------|-------------------|
| 0                          | 1,024           | 3,600                           | 0.4               |
| 1                          | 2,048           | 7,200                           | 1.4               |
| 2                          | 8,192           | 14,400                          | 2.5               |

3. *Results*

3.1. *Validation of Mesh Resolutions*

Prior to the analysis of computed results, it is important to confirm that the used mesh is able to resolve the turbulent boundary layers of our interest. The smallest active eddies in the turbulent boundary layers are the streamwise vortices with a diameter approximately of $30 \cdot l$, a streamwise length approximately of $300 \cdot l$ and spanwise spacing approximately of $150 \cdot l$ where $l$ is the viscous...
length scale defined as the ratio of the kinematic viscosity against the local friction velocity. To accurately resolve the turbulent boundary layer, the computation spatial discretisation should be smaller than these values. If this is not met, the accuracy will decrease.

By using computed results without mesh refinements, the wall shear stress on the rotating shaft was calculated. Using these results, viscous length scale was estimated and compared to the spatial discretisation of the computational mesh (Table 2).

Table 2. Mesh Discretisation Validation.

| Case | \( v^* \times 10^7 \) | \( Re_z \) | \( Re_o \) | \( u_t^* \times 10^6 \) | \( l^* \times 10^6 \) | Resulting Mesh Discretisation |
|------|----------------|-----------|-----------|----------------|----------------|-----------------------------|
| 1    | 4.040          | 110       | 450       | 0.0234         | 17.3           | 3 - 8 - 20                  |
| 2    | 2.020          | 220       | 900       | 0.0201         | 10.1           | 5 - 14 - 35                 |
| 3    | 1.010          | 440       | 1800      | 0.0177         | 5.7            | 9 - 25 - 62                 |
| 4    | 0.505          | 880       | 3600      | 0.0158         | 3.2            | 15 - 44 - 109               |

In Table 2:
- \( v^* \) is the kinematic viscosity normalised by reference velocity and length;
- \( Re_z \) is the Reynolds number based on the axial velocity of the flow and the clearance between the shaft and the casing;
- \( Re_o \) is the Reynolds number based on the shaft circumferential velocity and the clearance between the shaft and the casing;
- \( u_t^* \) is the friction velocity normalised by the reference velocity;
- \( l^* \) is viscous length scale normalised by the reference length;
- \( \Delta r, \Delta \theta \) and \( \Delta z \) are the mesh radial, circumferential and axial discretisation, respectively.

It can be confirmed that the mesh discretisation will be sufficient because it has several elements to compute streamwise vortices if they existed after two refinements. Based on these results, it was decided to perform the computations for cases \( Re_o = 450 \) and 900 with one refinement and cases \( Re_o = 1800 \) and 3600 with two refinements. Hereafter, only results using one refinement for \( Re_o = 450 \) and 900, and two refinements for \( Re_o = 1800 \) and 3600 will be shown.

3.2. Velocity Profile

The computed velocity profile for the axial component is presented in Figure 3. These velocity profiles were obtained in the liner part from the data averaged during one shaft revolution. It can be observed that for the two smallest Reynolds numbers, the velocity profiles are almost identical and clearly are of laminar.
When the Reynolds number is increased to 1800, the velocity profile slightly deviates from the one at $Re_w=450$ and 900. However, it is still close to the laminar profile. At the highest Reynolds number, the velocity profile is clearly of turbulent.

3.3. Visualisation of Flow Structures
To observe flow structures, iso-surface of positive $Q$-criterion coloured by vorticity magnitude $\Omega$ are presented in Figure 4 for three consecutive grooves. Figure 5 presents the same structures, focussing on one groove at different radius and on a cutting plan at a given circumferential location. These figures were obtained by averaging flow field over one shaft revolution.

![Figure 4. Iso-Surface of Positive $Q$ Criterion Coloured by Vorticity Magnitude.](image)
Figure 5. Positive $Q$-Criterion at Various Radial Locations and on Cutting Plan

In the second half of the grooves, for all the Reynolds numbers, a large recirculating region is visible. The size of this recirculation increases when the Reynolds number increases from $Re_\omega=450$ to 900 and for the higher Reynolds numbers its size seems unchanged. In the first half of the groove, two smaller recirculations are visible for all cases although they are hardly seen for the upper one at $Re_\omega=450$.

For $Re_\omega=450$ and 900, two recirculating regions can be observed just downstream of the inlet of the groove and just upstream of the exit. They are close to the shaft, in the clearance between the shaft and the groove. These two recirculating regions gradually vanished with increasing Reynolds number. This is confirmed by analysing the $Q$-criterion on a given radius near the shaft as presented in Figure 5.

In the liner part between each groove (recall Figure 1), the two recirculating regions located near the casing (stationary wall) are almost not affected by the Reynolds number. This is not the case for the one located near the shaft (rotating wall) just downstream of the groove’s exit, which disappear for $Re_\omega=1800$. This corresponds to the fact that the streamwise vortices start to develop from $Re_\omega=1800$.

When the streamwise vortices start to develop, they are scattered in the liner part. Additionally, larger vortices are visible in the groove under the main recirculation. For the highest Reynolds number
(\text{Re}_\omega=3600), the streamwise vortices are fully visible in the liner part, but also in the groove under the main recirculating region.

### 3.4. Head Losses

To quantify the effectiveness of the inter-stage bush design, the total losses of the bush \(\Delta\psi\), calculated as the total pressure difference between the inlet and the outlet of the bush, have been decomposed into two parts and they are presented in Figure 6: the losses in the liner part of the bush (between grooves, circles in Figure 6) and the losses due to the grooves (triangles in Figure 6).

#### Figure 6. Losses in Liner Part (Green Circles) and in Groove Part (Red Triangle) of the Inter-Stage Bush.

It can be observed in that figure that the losses due to the liner decreases with increasing Reynolds number. This trend is similar to the one for the flow in an inter-stage bush without groove [10]. However, for a grooved liner it is not possible to accurately predict losses by using empirical formula due to the different recirculation in the liner part (recall Figure 4 and Figure 5).

Regarding the losses caused by the presence of grooves, the trend is totally different. It can be observed that the groove losses increase from \(\text{Re}_\omega=450\) to 900. For higher Reynolds numbers, the losses decrease. For the sake of clarity, one can conclude that the effectiveness of the groove to generate losses and so to limit leakage flow decreases at high Reynolds number.

To increase the losses due to the groove at a high Reynolds number, new bush needs to be used with different groove design or an increased number of grooves.

### 4. Conclusions

The aim of this research is to investigate the flow fields and resulting pressure losses in grooved inter-stage bush of a centrifugal pump. For that purpose, fully resolved LES, which computes directly the turbulent boundary layer, was performed for different Reynolds numbers. The spatial discretisation of the computational mesh was checked in a preliminary computation to guarantee sufficient grid resolution for the active eddies in the turbulent boundary layer if they existed.

Main findings of the present research can be summarised as follow:

- Flow in the groove is mainly composed of one big recirculation in the second half of the groove and of two small recirculations near the inlet of the groove. Additionally, in the groove near the shaft, recirculations are visible at low Reynolds number, before they vanish gradually when the Reynolds number is increased;
- Despite the fact that the axial velocity profile is close to the laminar profile, streamwise vortices of the turbulent boundary layer started to be observed from \(\text{Re}_\omega=1800\) and were fully developed when Reynolds number was further increased;
Based on the above-mentioned observation, it is then needed to apply fully resolved LES for \( Re_\omega = 1800 \) or higher. For such a case, the increase in the number of computational elements would become significant;

Finally, based on the analysis of the losses in the stage bush, it was observed that the present design effectiveness to generate losses and so to limit leakage flow decreases for \( Re_\omega \) over 900. A different design should then be used.

**Nomenclature**

- \( C_0 \): reference velocity
- \( D_2 \): impeller outlet diameter
- \( D_s \): shaft diameter at stage bush
- \( e \): clearance at bush between shaft and casing
- \( g \): acceleration due to gravity
- \( H \): pump total head
- \( l \): viscous length scale, \( l = \frac{v}{u_\tau} \)
- \( l^* \): normalised viscous length scale, \( l^* = \frac{l}{L_0} \)
- \( L_0 \): reference length
- \( Q \): pump flow rate
- \( Re_z \): Reynolds number (based on axial velocity)
- \( Re_\omega \): Reynolds number (based on shaft circumferential velocity), \( Re_\omega = \frac{0.5 \cdot D_s \cdot \omega \cdot e}{v} \)
- \( u_\infty \): freestream velocity of the boundary layer
- \( u_\tau \): friction velocity, \( u_\tau = \sqrt{\frac{\tau_w}{\rho}} \)
- \( u_\tau^* \): normalised friction velocity, \( u_\tau^* = \frac{u_\tau}{C_0} \)
- \( U_2 \): impeller outlet peripheral velocity, \( U_2 = \omega \cdot \frac{D_2}{2} \)
- \( v \): kinematic viscosity
- \( v^* \): normalised kinematic viscosity, \( v^* = \frac{v}{L_0 \cdot C_0} \)
- \( \rho \): density of working fluid
- \( \tau_w \): wall shear stress
- \( \psi \): total pressure coefficient
- \( \omega \): pump shaft angular velocity
- \( \omega_s \): universal specific speed, \( \omega_s = \frac{\omega \cdot Q^{0.5}}{(g \cdot H)^{0.75}} \)
- \( \Omega \): vorticity magnitude
- \( \Omega^* \): normalised vorticity magnitude, \( \Omega^* = \frac{\Omega}{u_\tau^*} \)

**References**

[1] Smagorinsky J. 1963 General Circulation Experiments with the Primitive Equations *Monthly Weather Review* Vol. 91 N. 3

[2] Germano M., Piomelli U., Moin P. and Cabot W. H. A Dynamic Subgrid-Scale Eddy Viscosity Model *Physics of Fluids* 3, 7 pp. 1760-1765

[3] Lilly D. K. A Proposed Modification of the Germano Subgrid-Scale Closure Method *Physics of Fluids* 4, 3 pp. 633-635

[4] www.ciss.iis.u-tokyo.ac.jp/rss21

[5] Kato C., Kaiho M. and Manabe A. 2003 An Overset Finite-Element Large-Eddy Simulation Method With Applications to Turbomachinery and Aeroacoustics *Trans. of the ASME Vol. 70* pp. 32-43
[6] Nagahara T. and Inoue Y. 2009 Investigation of Hydraulic Design for High Performance Multi-Stage Pump Using CFD Proc. of the ASME 2009 Fluids Engineering Division Summer Meeting (Vail, Colorado)

[7] Pacot O., Kato C. and Avellan F. 2014 High-Resolution LES of the Rotating Stall in a Reduced Scale Model Pump-Turbine Proc. of the 27th IAHR Symposium on Hydraulic Machinery and Systems (Montreal)

[8] wci.llnl.gov/simulation/computer-codes/visit

[9] Prunières R., Inoue Y. and Nagahara T. 2016 Investigation of the Flow Field and Performances of a Centrifugal Pump at Part Load Proc. of the 28th IAHR Symposium on Hydraulic Machinery and Systems (Grenoble)

[10] Yamada Y. 1962 Resistance of a Flow Through an Annulus With an Inner Rotating Cylinder Bulletin of JSME Vol. 5 Issue 18 pp. 302-310