Front-Loading in the Simulation-Driven Design Process of Steam Turbine Blades Using Hybrid Computational Method of Quasi and Fully 3D Fluid Simulation*

Yixiang FENG**, Shigeki SENOO*** and Yoshio SHIKANO***

** Mechanical Engineering Research Laboratory, Hitachi Ltd., 832-2, Horiguchi, Hitachinaka, Ibaraki 312-0034, Japan
E-mail: yixiang.feng.cq@hitachi.com

*** Energy and Environmental Systems Laboratory, Hitachi Ltd., 7-2-1, Omika, Hitachi, Ibaraki 319-1221, Japan

Abstract

We developed a hybrid computational method for the design of steam turbine blades. It is composed of three main blocks: the meridian simulation, 3D simulation, and mixing plane blocks. This method was applied to a one-stage medium-pressure turbine and a four-stage low-pressure steam turbine. The results showed that the computation time was approximately half or even less than that of a fully 3D simulation. There was good agreement of the main design parameters, which suggests that our method is useful for the design of turbine blades. By use of this hybrid simulation method, front-loading of the blade design can be achieved and the number of feedbacks in the design process can be reduced. We believe that the proposed method is potentially useful for the simulation-driven design of various industrial products.

Key words: Steam Turbine, Multistage, Computational Fluid Dynamics, Blade Design, Hybrid Simulation, Through-Flow Simulation, Front-Loading, Simulation-Driven Design

1. Introduction

Recently, as symbolized by the opening of the Fifteenth Session of the Conference of Parties to the United Nations Framework Convention on Climate Change (COP15)(1), consideration of environmental factors such as CO₂ emissions and global warming, has become a major issue that all countries and governments have to deal with. As one of the key machines in electric power generation, efficiency of steam turbines has a large impact on global CO₂ emissions. Therefore, there is a strong need to improve their efficiencies.

The rotating blades (also called rotors or buckets) in steam turbines are the most important component, since they are directly responsible for transferring thermal energy of steam to rotational energy. Stationary blades (also called stators or nozzles) are located after the rotating blades and are also important to further decrease the pressure of the steam and to divert the flow angle to match the inlet angle of the next stage of the production process. Large power steam turbines typically go through more than 20 stages to fully collect the energy of pressurized steam generated in boilers. Therefore, careful design of the rotating blades for all stages to increase the total efficiency of the steam turbine is an important yet challenging for steam turbine manufacturers.

Due to the rapid development of computer hardware and numerical methods,
computational fluid dynamics (CFD) is now playing a vital role in the aerodynamic design of steam turbine blades\(^{(2)(3)(4)}\). Various CFD methods are used to meet the requirements of different stages in the design process of steam turbine blades. For example, the combination of through-flow calculation and blade-to-blade calculation is a quasi-3D method that adopts the axisymmetric assumption. The merit of quasi-3D methods is the short calculation time. However, since the detailed blade shape distribution in the circumferential direction is not considered, quasi-3D methods cannot be used for the detailed 3D design of blades. Therefore, quasi-3D methods are generally used in the pre-design stage. To consider the actual 3D effects of the blades, a fully 3D calculation is needed. Multistage calculation is also necessary to consider a real machine environment. However, despite the increasing speed of supercomputers nowadays, the time taken for such fully 3D multistage calculation is still too long for optimal design process.

Therefore, to achieve a solution compromising both high calculation speed and accuracy, we chose a hybrid computational method that combines multistage quasi-3D through-flow calculation and fully 3D calculation of the rotor. The quasi-3D through-flow calculation determines the upstream and downstream boundary conditions for the 3D calculation. In addition, the thermodynamic properties of steam are taken into account for the two-phase gas-liquid flow in low-pressure blades.

The concept of hybrid simulation for turbine cascades was first reported by Dawes and Denton\(^{(10,11)}\). However, very little research has been conducted on the industrial application of such hybrid simulation methods\(^{(14)}\). One reason is that many designers use commercial CFD software in which such hybrid simulation is not readily available. In this paper, we report the development of a hybrid computational fluid simulation method using in-house-developed CFD and mesh generation codes. Application of this method to the CFD calculation of several types of steam turbine blades is also reported. We also discuss the possibility of using this hybrid method to shorten the time required for the design of steam turbine blades.

2. Nomenclature

- MP: mixing plane at interface of through-flow calculation and 3D calculation
- p: static pressure
- \(v\): velocity
- \(Q\): flow rate
- \(h\): enthalpy
- \(\rho\): density
- \(e\): specific total energy
- \(\varepsilon\): specific inner energy
- \(\mu\): dynamic viscosity
- \(\omega\): rotational speed
- \(F\): blade force
- \(H\): specific total enthalpy
- \(M\): Mach number
- \(T\): deformation tensor
- \(R\): Reynolds stress tensor
- \(\Omega\): control volume
- \(n_j\): normal unit vector with respect to surface in calculation

3. Subscripts

- \(x\): streamwise (axial) direction
- \(r\): radial (blade height) direction
- \(\theta\): tangential (circumferential) direction
4. Design Process of Steam Turbine Blades

A typical design process for steam turbine blades is shown in Fig. 1. First, the basic specifications such as the number of stages and the total power output are designed. Then, an efficient one-dimensional in-house-developed code is used to determine the general thermal and fluid condition of each stage. After the flow condition at the meridian plane is confirmed, 2D design of the blades in each stage is performed, based on the triangular relationship of the velocity components. Then, 2D blade-to-blade simulation is carried out to confirm the flow conditions at each section. After this, “stacking” is performed to generate 3D blades. In the stacking process, the shape of the blades at each cross section in the radial direction is designed. Stress and vibration analysis is performed so that the 3D stator becomes available for evaluation of the 3D rotor. Based on the results of the blade-to-blade simulation, stacking, stress and vibration analysis, and fully 3D simulation, feedback to the 2D design of the blade cross section is performed if necessary. After all the feedback is performed and an analysis of the fully 3D simulation shows that the performance of the steam turbine is satisfactory, experimental analysis is conducted using test rigs.

As shown in Fig. 1, the time for blade design depends largely on the number of feedbacks required by the system. Therefore, to reduce the feedback time, front-loading of the 3D rotor evaluation is necessary. An ideal tool is a simulation of the newly stacked 3D rotor using 2D stators. Therefore, the development of a hybrid simulation method can not only decrease the simulation time of the 3D calculation, but also enable front-loading of the 3D evaluation.

5. Methods

5.1 Hybrid simulation

A typical hybrid simulation system contains three blocks. (1) a through-flow calculation block (also named a “meridian simulation” because the calculation is performed at the meridian plane), (2) a 3D calculation block, and (3) a mixing plane block, which exchanges boundary conditions between the through-flow calculation and the 3D calculation. The basic structure of the hybrid simulation is shown in Fig. 2.
A flowchart of the hybrid simulation system is shown in Fig. 3. Shape and boundary conditions are read from a text file, where the stage number of the 3D calculation is also designated. The mesh data for the 3D stage is read separately from a mesh file. After this setup, meridian region 1, which is upstream of the 3D stage, is calculated using an in-house through-flow solver for one iteration. The results at the exit of meridian region 1 are transferred to the inlet of the 3D region as the inlet boundary condition. Then, the flow in the 3D stage is calculated for one iteration using an in-house 3D CFD solver. The results at the exit of the 3D stage region are transferred to the inlet of meridian region 2, which is located downstream. The flow in meridian region 2 is then calculated for one iteration. Convergence is judged from the mass flow of all the regions. The whole process will repeat until the convergence criterion is satisfied.

5.2 Through-flow calculation

The through-flow calculation uses a quasi 3D flow analysis. It solves the compressible flow equation in the meridian plane using axisymmetric cylindrical coordinates. The forces acting on the blades are calculated from the camber and lean angles of the blades using the boundary conditions on the blades. The governing conditions are written in the tensor form below.

\[
\frac{\partial}{\partial t} \int_{\Omega} Q d\Omega + \int_{\partial\Omega} (E_j \cdot n_j) dl + \int_{\partial\Omega} S d\Omega + \int_{\Omega} F d\Omega = 0
\]

\[j = x, r\]

Here, \(n_j\) is the normal vector with respect to the grid line in the meridian plane.

\[n_j = (n_x, n_r)\]  

The matrix \(Q, E, S,\) and \(F\) are defined as follows.
In Eq. 2, \( e \) is the specific total energy defined as follows.

\[
e = \rho \cdot \left[ \frac{1}{2} (u_i^2 + u_j^2 + u_k^2) \right] = \rho h + \frac{1}{2} \rho \cdot (u_i^2 + u_j^2 + u_k^2)
\]

(4)

\( F \) is the forces acting on the steam due to the blades and has the following boundary conditions. For a stationary nozzle, it is

\[
F_i u_i^* + F_i u_i + F_i u_0 = 0
\]

(5)

For a rotor, it becomes

\[
F_i u_i^* + F_i u_i + F_i (u_0 - r \omega) = 0
\]

(6)

Equation 1 is solved by using a FVM (Finite Volume Method) based on the third-order upwind TVD (Total Variance Diminishing) scheme. To increase the accuracy of the numerical model, a generalized approximate Riemann solver has been developed. In addition, the thermodynamic properties of the steam are considered using a steam table matrix. More detailed description of the numerical technique used in this study is described in references (3-5).

### 5.3 Three-dimensional calculation

In the fully 3D calculation, the governing equations of mass, momentum, and energy are solved. The fluid is considered to be a mixture of liquid and gas. The governing equations can be written as follows.

\[
\frac{\partial}{\partial t} \int_\Omega Q d\Omega + \int_{\partial \Omega} (E_j \cdot n_j) dS - \int_\Omega (F_{ij} \cdot n_j) dS + \int \mathcal{J} d\Omega = 0
\]

(7)

Here the matrix \( Q, E, F, \) and \( J \) are defined as follows.

\[
Q = \begin{bmatrix}
\rho \\
\rho u_i \\
\rho u_j \\
e
\end{bmatrix},
E_j = \begin{bmatrix}
\rho u_i \\
\rho u_i u_j + p \delta_{ij} \\
\rho u_i u_j + p \delta_{ij} \\
(e + p) u_j
\end{bmatrix},
F_{ij} = \begin{bmatrix}
0 \\
\mu T_{ij} - R_{ij} \\
\mu T_{ij} - R_{ij} \\
\delta_{ij}
\end{bmatrix},
J = \begin{bmatrix}
0 \\
\rho \omega^2 y + 2 \rho \omega u_i \\
\rho \omega^2 z - 2 \rho \omega u_i \\
\rho \omega^2 (u_i y + u_i z)
\end{bmatrix}
\]

(8)

Here, \( T_{ij} \) is the deformation tensor defined as follows.

\[
T_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_i} \delta_{ij}
\]

(9)

where \( \delta_{ij} \) is the Kronecker delta.

\( R_{ij} \) is the Reynolds stress tensor defined as

\[
R_{ij} = -\rho u_i u_j
\]

(10)

\( \mathcal{G}_j \) is the thermal flux defined as

\[
\mathcal{G}_j = \lambda \frac{\partial T}{\partial x_j} + \mu T_{ij} u_i - \rho h u_j
\]

(11)

Finally, \( n_j \) is the normal unit vector with respect to the calculating surface.
5.4 Boundary condition exchange at mixing plane
The mixing plane is the connecting block between the through-flow calculation and the 3D calculation and therefore plays an important role in the hybrid simulation system. Definition of the two mixing planes MP1 and MP2 has been shown in Fig. 2. The exchange of boundary conditions is performed in such a way that the mass, momentum, and energy across the mixing plane are conserved. The details of the boundary condition exchange at MP1 are shown in Fig. 4. Boundary condition exchange at MP2 can be inferred in a similar way.

Details of the boundary condition exchange at the mixing plane are described below.

(1) MP1 at interface between upstream through-flow calculation and 3D calculation. In this MP, total pressure, total enthalpy, and flow angles are transferred from the upstream through-flow to the 3D calculation at each radial direction. The circumferential distribution of these parameters is assumed to be uniform and these parameters are used as inlet boundary conditions for the 3D calculation. At the same time, the static pressure at the inlet of the 3D calculation zone is averaged circumferentially and transferred to the upstream through-flow zone. This is used by the through-flow calculation as a boundary condition at the outlet.

(2) MP2 at interface between 3D calculation and downstream through-flow calculation. In this MP, velocities and specific total energy are circumferentially averaged and transferred to the downstream through-flow calculation at each radial direction. These parameters are used as the inlet boundary condition for the downstream through-flow calculation. At the same time, the static pressure at the inlet of the downstream through-flow calculation zone is transferred to the 3D calculation zone. The circumferential distribution of the static pressure is assumed to be uniform. The static pressure is used as an outlet boundary condition for the 3D calculation. Although interpolation of parameters is possible at the radial directions MP, we choose to exchange the boundary conditions directly by forcing the grid generation to be identical at the radial direction, in order to avoid any interpolation errors.

6. Application
6.1 Application to Medium-Pressure Turbine
The developed hybrid simulation method was first applied to a one-stage calculation of a medium-pressure turbine blade. The specification of the medium-pressure turbine is shown in Table 1. The pressure ratio is defined as the ratio of pressure between the outlet and the inlet. The number of meshes for the through-flow calculation, hybrid simulation, and fully 3D calculation were 18,000, 315,000, and 601,920, respectively.
Table 1 Specification of the medium-pressure turbine

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotational velocity</td>
<td>3,000 rpm</td>
</tr>
<tr>
<td>Number of rotating blades</td>
<td>76</td>
</tr>
<tr>
<td>Number of stationary blades</td>
<td>74</td>
</tr>
<tr>
<td>Pressure ratio</td>
<td>0.86</td>
</tr>
</tbody>
</table>

The simulation was performed on a PC (AMD Opteron 2.8G, 1CPU). The calculation time for the fully 3D calculation was approximately 24 hours, for the through-flow calculation was about 4 hours, and for the hybrid simulation was approximately 12 hours. Therefore, it was confirmed that the calculation time of the hybrid simulation was about half that of the fully 3D calculation.

The distribution of the degree of reaction along the height of the rotors was shown in Fig. 5. The degree of reaction represents the distribution of blade load on the rotating blade and is one of the most important parameters in the design of turbine blades. The through-flow calculation is indicated by “meridian” in the figure. The ordinate is the radial position of the blade normalized by the blade length. The degree of reaction is defined using the following equation.

\[ R_s = \frac{h_{s1} - h_{s2}}{h_{0in} - h_{s2}} \times 100\% \]  \hspace{1cm} (12)

Here, \( h_{s1}, h_{s2} \) and \( h_{0in} \) represent static enthalpy at the inlet of the rotating blade, static enthalpy at the outlet of the rotating blade, and total enthalpy at the inlet of the stationary blade, respectively.

Figure 5 shows that the inclusion of 3D calculation of rotor in the hybrid simulation results in a large improvement over the 2D through-flow calculation. The hybrid simulation results agree very well with the 3D calculation in most of the region. The discrepancy is relatively large near the hub and tip area of the blade, which may be due to endwall effects. Therefore, the results show that our method is suitable for its design purpose, if special attention is paid to the hub and tip regions.

![Fig. 5 Comparison of degree of reaction](image)

The static pressure distribution at the midspan is compared in Fig. 6. Reasonably good agreement is can be observed between the hybrid simulation and the 3D simulation.
Since the high precision of the fully 3D simulation code has been validated by experimental studies, it can be inferred that the hybrid simulation has adequate accuracy for its design purpose.

6.2 Application to Low-Pressure Turbine Cascade

The newly developed hybrid simulation method was applied to the calculation of last four stages of a low-pressure steam turbine. The rotor of the third stage from the exit (L-2) was calculated in a fully 3D shape as shown in Fig. 7. The number of meshes for the upstream through-flow calculation (L-3), fully 3D calculation (L-2), and downstream calculation (L-1 and L-0) were 8,940, 337,550, and 17,280, respectively. The pressure ratio (ratio of pressure between the outlet and the inlet) for this calculation was 0.035.

The simulation was performed using a PC (AMD Opteron 2.8G, 1CPU). The calculation time for the hybrid simulation is approximately 1/2 as compared to the fully 3D calculation of L-2. The time saving effect will be more prominent if we compare the calculation time to fully 3D multistage simulation. It is also confirmed that the error of the steam mass flow rate across the mixing planes is less than 0.5%.

The distribution of typical design parameters along the height of the rotors was shown in Fig. 8. The through-flow calculation is indicated by “meridian” in the figure. The ordinate is the radial position of the blade normalized with respect to the blade length.
results of hybrid simulation showed some improvement over those of the through-flow calculation. Although there were still some discrepancies, especially at the tip region, they were limited and thus can be tolerated in the design process if special attention is paid to those regions.

7. Discussion

This paper introduces the development of a hybrid computational fluid dynamics model that combines 3D calculation with axisymmetric through-flow calculation. This model was applied to the calculation of medium- and low-pressure steam turbine cascades and was proved to give moderately accurate solutions for the designed blades.

(1) Time map for different simulation methods

On the basis of the results of the four-stage low-pressure turbine cascade, we can compare the time required for different simulation methods. A time map for different types of fluid simulations is shown in Fig. 9. To remove the effects of the number of stages and convergence, we define the “normalized calculation time”, which is the calculation time normalized by the time of the through-flow simulation. Through-flow simulation is the fastest one of all the methods and its “normalized calculation time” is 1. Although this may not be the general case for all turbine blades, it can give us a general idea about the scale of time needed for each calculation. The hybrid simulation can thus be inferred to have a good cost performance ratio. It should be noted that the actual calculation time depends on the number of stages, mesh numbers of each stage. It may also be dependent on the convergence of the solver, because the iterations needed for 3D calculation and for hybrid calculation can be different.
(2) Selection of mixing plane method

The methods for calculating multistage turbomachinery have been widely studied by many other researchers (12-14). These methods include successive analysis of isolated blade rows, mixing plane method, averaging-passage method, and fully unsteady method.

Our choice of the mixing plane method is based on the following advantages. (1) It is able to represent a multistage environment while solving the equation for an individual blade flow. (2) The inter-blade row boundary conditions are calculated rather than being imposed. (3) The cost performance of the mixing plane method is generally better than that of other methods.

According to the research of Wyss et al. and Chima, several schemes exist for averaging boundary conditions. These schemes include a mixed-out average, kinetic energy average, and area average scheme. An area average does not conserve mass and is the least useful of the three. The mixed-out average was used by Denton. It conserves mass, momentum, and energy. However, mixing losses may occur(10). In the present work, kinetic energy average method is used. It conserves mass and total enthalpy. The static enthalpy and velocity components are mass-averaged in this approach. Further details of the averaging method were described by Wyss et al. (12).

(3) Towards front-loading in simulation-driven design (SDD)

With the development of computer hardware and computational technology such as that for large-scale parallel computation, computer simulation is now playing a key role in the design of many industrial products. Although computer simulation still cannot fully replace experiments, it can effectively shorten the development cycle by minimizing the experimental tests necessary. The extensive use of computer simulation in innovative product design has led to the concept of “simulation-driven design (SDD)” otherwise known as “analysis-led design (ALD)” (15).

While SDD/ALD requires a large number of simulation tests for parameter survey, the computational time is still too costly in some cases, especially in CFD, where turbulence and some other non-linear and unsteady phenomena require fine spatial and temporal resolution. The time cost is even amplified by the need for feedback in the design process. Since feedback in the design process results in a repetition of the whole simulation, reduced feedback can effectively shorten the development cycle.

A typical product development process is shown in Fig. 10. Product design typically starts with preliminary or conceptual design, which is usually based on the needs of...
customers. After that, functional design is performed to determine the function of each component. Then, relationship between different components and the product as a whole are considered and this is the phase of detailed design. After this phase, candidates that performs well will be rapid prototyped to test and validate the design. In SDD/ALD process, simulation tools are used to aid the design of the product. As shown in Fig. 10, increase in the complexity of design is accompanied by the increase of dimensionality of the simulation. Therefore, time cost for simulation in later process increases and repetition of simulation should be reduced to a minimum level.

In this study, we have addressed this problem by using front-loading in the SDD/ALD process. Front-loading is used in the design and development process to shorten development lead times. It is defined as “a strategy that seeks to reduce development time and cost by shifting the identification and solving of design problems to earlier phases of a product development process” (16). In this paper, we have used the phrase “front-loading” in a broader sense, which indicate the shifting of simulation methods in the design process. This shifting can be achieved by changing the sequence of existing simulation methods or by introducing new simulation methods. In this study, the latter is used. With the help of the newly developed hybrid simulation, 3D evaluation of the rotating blade under design is front-loaded. Since extensive use of multi-scale and multi-physical simulation is typical in the SDD/ALD process, it can be inferred that this concept of front-loading is potentially useful for many other industrial products.

8. Conclusion

We have developed a hybrid computational fluid dynamics method for the aerodynamic design of steam turbine blades. It was applied to the calculation of medium-pressure and low-pressure turbine cascades. The computational time was reduced to approximately half of that of fully 3D calculation. There was good agreement in the distribution of major design variables, which guarantees the effectiveness of the hybrid simulation method in the design process. Therefore, we conclude that this method can be used to improve the efficiency of the design of steam turbine blades.

Taking advantage of the hybrid simulation method, we can construct an improved design approach for steam turbine blades. The performance of the 3D rotor after stacking can now be immediately evaluated using the hybrid simulation method. Compared with the conventional design process, which evaluates the 3D effect of the rotor using fully 3D calculation, our new process enables front-loading in the simulation-driven design process. It can reduce the number of feedbacks and results in a more efficient blade design process.
We conclude that this concept of front-loading introducing our new simulation method is potentially useful in the simulation-driven design of various industrial products.

References


