A Study of the Unsteady Flow Field and Turbine Vibration Characteristic of the Supersonic Partial Admission Turbine for a Rocket Engine

Yuki Tokuyama, Ken-ichi Funazaki, Hiromasa Kato, Noriyuki Shimiyama, Mitsuru Shimagaki and Masaharu Uchiumi

1 Department of Mechanical Engineering, Iwate University, 4-3-5 Ueda, Morioka, Iwate, 020-8551, Japan
2 Japan Aerospace Exploration Agency, 1, Kimigaya aza Koganezawa, Kakuda, Miyagi, 981-1525, Japan

ABSTRACT

Turbines used in upper stage engine for a rocket are sometimes designed as a supersonic turbine with partial admission. A turbine with partial admission nozzle causes strong unsteadiness in the flow field due to the existence of the closed nozzle passages. Although there are a number of published numerical studies on steam turbines employing partial admission nozzle, three dimensional CFD analysis of a supersonic partial admission turbine are rarely dealt with in those studies. In this study, a three dimensional unsteady CFD analysis of the supersonic partial admission turbine has been conducted in order to gain knowledge of three dimensional flow patterns, unsteady pressure variation and aerodynamic forces on the rotor blade. The result shows strong three dimensional flow pattern, such as large separation zone, in the region between closed nozzle passages and rotor blades. Furthermore, large unsteady aerodynamic forces appear on the rotor blade with the largest harmonic EO (Engine Order) component occurring at the nozzle closed sector passing frequency. This study also conducts one-way fluid-structural interaction (FSI) analysis in order to investigate turbine vibration response and cyclic stress. The unsteady pressure data on each grid points on the rotor blades obtained by the CFD analysis are transferred to the FEM model and unsteady nodal forces on each FEM nodes are calculated. FFT operations of these unsteady nodal forces are conducted and characteristic unsteady components, 3EO, 36EO and 45EO, are extracted on each FEM nodes. The 3EO force component is the largest one and corresponds to nozzle closed sector passing frequency. The others are force components which cross 1st or 2nd eigenfrequency curves of a rotor blade mode shapes around turbine design rotation speed on a Campbell diagram. Frequency response analyses of a rotor blade model and a turbine full angular model are conducted using those force components as nodal force conditions. The frequency response analyses of a rotor blade model indicate that 36EO and 45 EO force components excite 1st and 2nd mode shapes. However, 3EO force component, the largest force component, does not excite these mode shapes because the force frequency is enough far from the eigenfrequencies of a rotor blade mode shapes. The result of frequency response analysis of a turbine full angular model shows that 3EO force component excites 3ND mode shape if the force frequency close to the eigenfrequency of that mode shape. However, 1ND or 2ND mode shapes are not excited even if the force frequency is close to eigenfrequencies of these mode shapes.

INTRODUCTION

Turbines for a rocket engine are used for driving turbopump systems. The turbines used for lower stage engine, for example LE-7A, the first stage engine of the H-2A and H-2B rocket in Japan, are usually required to produce large output power. Thus, the turbines are supplied with working gas at flow rate sufficient for a full admission configuration. On the other hand, the output power required for upper stage turbines, for example LE-5B, the second stage engine of the H-2A and H-2B rocket, is not as large. So, if the working gas has enough energy, the turbines must be driven with small flow rate [1]. Thus, a turbine with a full admission configuration would result in turbine blades with extremely low blade height or the diameter must be much smaller to avoid extremely low blade height. The former design causes the increase in some kind of losses, for example passage vortices or tip leakage flow and friction etc., and the latter causes the decrease in velocity ratio. So, both designs result in a decrease of turbine efficiency. In order to prevent these undesirable features, a partial admission configuration can be employed [1]. In the partial admission configuration, several groups or sectors of nozzle inlets are completely closed and because of this the blade height and turbine diameter can be kept at a reasonable value. This configuration is actually used in LE-5B and considered in the next generation rocket engine as well. However, the existence of the closed nozzle sector creates additional losses such as windage loss, end off sector loss and expansion loss [2]. Furthermore, the rotor blades experience very unsteady aerodynamic force because they pass through open and closed nozzle passages periodically and this may cause high cycle fatigue (HCF) or fatal turbine vibration.

The partial admission turbine is also used in steam turbine control stages in order to control output power. Numerous studies focusing on aerodynamic performance, both experimentally and numerically, have been conducted in the past. Sakai, et al. [3] compared quasi-three-dimensional CFD analysis with experimental results. Three-dimensional unsteady CFD simulations are also conducted in recent years. Hushmandi, et al. [4] presented unsteady aerodynamic force on the rotor blades. Kalkkuhl, et al. [5] showed detailed flow patterns in a steam turbine control stage. Yoshida, et al. [6] compared unsteady pressure fluctuation on a rotor blade predicted by numerical approach including disk cavity domain model with unsteady data measured by experimental approach and the results showed good agreement with each other. Apart from aerodynamic point of view, a forced response analysis based on experimental approach of a partial admission turbine was conducted by Fridh, et al. [7]. In contrast, there are few cases treating supersonic turbine for a rocket. Aerodynamic experimental researches were conducted in Ref. [8,9] in 1970s and showed the influence of the number of the closed nozzle sector to the turbine efficiency. The authors conducted CFD analyses, in Ref. [10,11], to understand basic unsteady flow patterns in recent years. However, these numerical studies dealt with two-dimensional analysis. Thus, very few three-dimensional unsteady flow patterns through the supersonic partial admission turbine stage for rocket have been presented. Although the partial admission configuration for rocket turbine can keep the blade height within a reasonable value com-
pared to full admission case, the blade height often becomes less than 10mm. Therefore, three-dimensional flow phenomena may have a great impact on the flow fields and investigating these flow fields in detail is very important to design a turbine with higher aerodynamic performance. In contrast to the aerodynamic research, to the author’s best knowledge, no studies dealing with the influence of the unsteady aerodynamic force caused by the partial admission configuration to the turbine vibration response has been published in the past. However, understanding how unsteady force components excite which and how turbine vibration mode shapes is very important in avoiding turbine resonance or HCF problems during the design phase.

In this study, three-dimensional unsteady CFD simulation is conducted in order to investigate the three-dimensionality of the unsteady flow field in the supersonic partial admission turbine. Furthermore, this study conducts one-way fluid-structural interaction (FSI) analysis to research the turbine response to the unsteady aerodynamic force. Frequency response analyses of a turbine blade and a turbine full annular model are performed by FEM procedure. The simulations of a turbine blade model are performed to focus on the blade response with reasonable computational cost and the simulations dealing with the full annular model are conducted to research the turbine disk response.

The unsteady nodal forces on each FEM node on the rotor blade surfaces are interpolated from the CFD result. Then, the unsteady forces are transformed into frequency domain by FFT procedure and the characteristic force components are extracted and used as nodal force conditions of the frequency response analysis.

SIMULATED TURBINE STAGE

The turbine studied in this study is a scaled model of the FTP turbine for NASA M-1 engine [9]. This turbine is a two-stage supersonic turbine with partial admission having three closed nozzle sectors. Three-dimensional partial admission CFD simulation needs large computational resources. The previous study [11], dealing with two-dimensional two-stage CFD simulation, indicated that the most complex characteristic and influential flow events caused by the nozzle closed sector, such as strong shock waves due to the flow acceleration, appeared in the first stage. Thus, only the first stage is treated in this study to reduce the computational cost. There are 44 nozzle passages and 24 nozzle passages are used as admitted sector and 20 nozzle passages are closed. The 1st stage rotor consists of 94 blades. The turbine blade profiles and characteristics are shown in Figure 1 and Table 1, respectively. In the simulation, blade scaling procedure is employed to reduce CFD simulation cost by using the periodic boundary condition as in Ref. [11]. So, the number of the each nozzle and rotor blade used in the calculation is changed to 45 from 44 and changed to 93 from 94, respectively. As a result of the scaling, one third of full passages are simulated, that is, 8 admitted and 7 closed passages for the nozzle and 31 passages for the 1st stage rotor, respectively, as shown in Figure 2. Detailed information about the scaling procedure is described in Ref. [11].

CFD PROCEDURE

URANS simulation is performed by ANSYS CFX 13 in this study. Temporal discretization is the second-order backward Euler method and the convective terms are discretized by a high resolution TVD scheme. The shear stress transport (SST) model is used as turbulence model, however, as mentioned later, wall-function treatment is applied globally near the wall. Simulated conditions correspond to the experimental conditions in Ref. [10] and total pressure, total temperature and flow angle are imposed on the inlet boundary and static pressure is imposed on the outlet boundary. The transient time step is 1/40 of the nozzle pitch passing period and this division number is of a comparable order to the simulation in Ref. [4, 5]. As mentioned above, one third of full passages are simulated using periodic boundary condition. The CFD simulation domain and detailed boundary conditions are summarized in Figure 2 and Table 2, respectively. In addition to the three-dimensional analysis, two-dimensional simulation on mean diameter is also conducted in this study to compare with the three-dimensional analysis. The two-dimensional simulation is conducted by the domain modeled with the 5% span height of the actual height and by treating the hub and shroud walls as slip wall.

![Fig. 1 Blade profiles of M-1 turbine for 1st stage](image1)

![Fig. 2 Computational domain for CFD simulation](image2)

<table>
<thead>
<tr>
<th>Table 1 Turbine blade characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nozzle</td>
</tr>
<tr>
<td>Number of blades (Before the scaling)</td>
</tr>
<tr>
<td>Axial chord length [mm]</td>
</tr>
<tr>
<td>Blade height [mm]</td>
</tr>
<tr>
<td>Tip clearance [mm]</td>
</tr>
<tr>
<td>Mean diameter [mm]</td>
</tr>
</tbody>
</table>

![Table 2 Numerical conditions](image3)

<table>
<thead>
<tr>
<th>Working fluid</th>
<th>N2: ideal gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotational speed [RPM]</td>
<td>10,080</td>
</tr>
<tr>
<td>Inlet conditions</td>
<td></td>
</tr>
<tr>
<td>Total pressure [MPa]</td>
<td>2.35</td>
</tr>
<tr>
<td>Total temperature [K]</td>
<td>273</td>
</tr>
<tr>
<td>Inflow angle</td>
<td>Normal to boundary</td>
</tr>
<tr>
<td>Outlet condition</td>
<td>Static pressure [MPa] 0.645</td>
</tr>
<tr>
<td>Wall condition</td>
<td>No slip and adiabatic</td>
</tr>
</tbody>
</table>
Prior to the partial admission simulation, grid independency is checked by steady simulation of one nozzle passage and two rotor passages using periodic boundary condition, that is, full admission analysis. Although the most desirable approach of grid independency test in this study is to simulate the flow with partial admission, the partial admission analysis requires a large amount of computational cost which is not realistic. Therefore, the grid independency test is conducted by the full admission simulation. Four different grids, Coarse, Middle, Fine and No wall-func. are generated by NUMECA AutoGrid 5 and compared each other. Although grid independency test is performed using SST turbulence model, CFX solver automatically switches the near-wall treatment from wall-functions to a low-Re turbulence model when the wall mesh spacing is sufficiently fine [12]. In this grid independency test, Coarse, Middle and Fine grid systems use wall-function treatment everywhere near the wall and the low-Re model is used in the No wall-func. grid case. It should be noted that Coarse and No wall-func. grids have almost the same grid resolution except near wall region. Thus, we can evaluate the influence of using wall function by comparing the results calculated by these two grid cases. Detailed grid information is summarized in Table 3.

Figure 3 shows turbine total-to-total efficiency and Figure 4 indicates spanwise axial velocity distributions at the nozzle exit and yaw angle distributions at the exit of the rotor passage simulated by the Coarse, Middle and Fine grids. We can see that the difference of the turbine efficiency among three grids is very small and falls within 0.3%. On the other hand, quantitative difference of axial velocity appears at 0–30% span region and yaw angle appears at 80–90% span region. However, the Coarse grid can sufficiently capture the qualitative distribution features in these region and obtain good agreement at the other span regions. Figure 5 is a comparison of static pressure distribution on the rotor blade mid-span section between the Coarse and No wall-func. grid cases. Although quantitative values are slightly different at 5–30% Cx on the suction surface, pressure distributions indicate good agreement between these two grids despite using wall-function treatment in the Coarse grid case. Based on these results, the partial admission analysis is conducted using almost the same grid resolution with the Coarse grid. Then, the global number of grid points used in the partial admission analysis is approximately twenty million and the computational grid is shown in Figure 6.

Table 3 Number of grid points and resolution used in the grid independency test

<table>
<thead>
<tr>
<th>Number of grid points</th>
<th>Coarse</th>
<th>Middle</th>
<th>Fine</th>
<th>No wall-func.</th>
</tr>
</thead>
<tbody>
<tr>
<td>per one passage</td>
<td>430,000</td>
<td>1,500,000</td>
<td>4,500,000</td>
<td>750,000</td>
</tr>
<tr>
<td>Nozzle</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotor</td>
<td>520,000</td>
<td>1,500,000</td>
<td>4,500,000</td>
<td>1,200,000</td>
</tr>
<tr>
<td>Number of grid points in O-mesh</td>
<td>17</td>
<td>25</td>
<td>25</td>
<td>29</td>
</tr>
<tr>
<td>Nozzle</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotor</td>
<td>13</td>
<td>21</td>
<td>25</td>
<td>29</td>
</tr>
<tr>
<td>Cell width at solid walls [m]</td>
<td>5.0E-06</td>
<td>5.0E-06</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Average y value on the wall</td>
<td>Approx. 0.2</td>
<td>Approx. 0.2</td>
<td>Approx. 0.2</td>
<td>Approx. 0.2</td>
</tr>
</tbody>
</table>

Fig. 3 Grid dependency check in terms of total-to-total efficiency

Fig. 4 Grid dependency check in terms of axial velocity at the nozzle exit and relative flow angle at the turbine exit

Fig. 5 Comparison of static pressure distributions on the rotor blade mid-span section calculated by using wall-function and low-Re model near the wall boundary

Fig. 6 Computational grid used in the partial admission analysis
FSI ANALYSIS PROCEDURE

Turbine vibration response to some frequency components of the unsteady aerodynamic force are investigated by one-way FSI approach. Commercial FEM software, MSC NASTRAN, is used for simulating the vibration response. Figure 7 is the flow chart of the FSI simulation in this study. The FSI simulation consists of three steps as shown in Figure 7. The first and third steps are simulating unsteady flow field and vibration response by CFD and FEM analysis, respectively. The second step is to transfer unsteady force from CFD to FEM model and this mapping process is conducted by in-house mapping program.

Fig. 7 A flow chart of one-way fluid-structural analysis conducted in this study

A concept of the mapping procedure in this study is to conserve the aerodynamic force between before and after the mapping process. The force can be conserved by using Eq. (1) [13].

\[ f_j = \sum_{i=1}^{j_1} \Phi_i c_{ij}, \]  

where, \( f_j \) is the transferred nodal force at the structural node \( j \), \( \Phi_i \) is the force at the CFD grid point \( i \), \( j_1 \) is the number of CFD grid points which will transfer its force to the structural node \( j \) and \( c_{ij} \) is interpolation coefficient. Furthermore, \( c_{ij} \) is determined so as to satisfy Eq. (2).

\[ \sum_{j=1}^{j_1} c_{ij} = 1, \]  

where, \( j_1 \) is the number of FEM nodes which will receive the force from the CFD grid point \( i \) and this number is 3 in this study as mentioned later.

In this study, FEM model consists purely of tetrahedral elements. A schematic of the force transfer from a CFD grid point to structural nodes are illustrated in Figure 8. All CFD grid points on the rotor blades are projected on the nearest surface cells of the FEM elements and a CFD grid point \( i \) will transfer its force to the structural 3 nodes which construct a surface cell of the projected element. Then interpolation coefficient \( c_{ij} \) is determined based on barycentric method of triangle as follows. Generally, the barycentre of the triangle which has different mass points at each three vertices is formulated as Eq. (3).

\[ \vec{G} = \frac{M_A \vec{A} + M_B \vec{B} + M_C \vec{C}}{M_A + M_B + M_C}, \]  

where, the vector \( \vec{G} \) is a position vector of the barycenter of the triangle, the vector \( \vec{A}, \vec{B} \) and \( \vec{C} \) are position vectors of each vertexes, the \( M_A, M_B \) and \( M_C \) are masses at each vertex. In order to determine the interpolation coefficient \( c_{ij} \), we replace the vector \( \vec{G} \) by vector \( \vec{Q} \) corresponding to the position vector of the projected point of the CFD grid point \( i \). Furthermore, we also replace the \( M_A, M_B \) and \( M_C \) by \( f_A, f_B, f_C \) corresponding to forces which the FEM node \( A, B \) and \( C \) will receive from the CFD grid point \( i \). Then, the Eq. (3) can be written as Eq. (4). It should be noted that the \( f_A, f_B, f_C \) must satisfy Eq. (5).

\[ \vec{Q} = \frac{f_A \vec{A} + f_B \vec{B} + f_C \vec{C}}{f_A + f_B + f_C}. \]  

\[ \Phi_i = f_A + f_B + f_C. \]  

Therefore, the Eq. (4) indicates that barycenter of the transferred forces on the three structural nodes corresponds to the projection point of the CFD grid point. Then, the Eq. (4) can be deformed and written in matrix form as Eq. (6).

\[ \begin{bmatrix} x_A & x_B & x_C \\ y_A & y_B & y_C \\ z_A & z_B & z_C \end{bmatrix} \begin{bmatrix} f_A \\ f_B \\ f_C \end{bmatrix} = \Phi_i \begin{bmatrix} x_Q \\ y_Q \\ z_Q \end{bmatrix}. \]  

where, \( x, y \) and \( z \) are Cartesian coordinates of each structural nodes and the projection point of the CFD grid point. Thus, the transferred forces on the three structural nodes can be determined by Eq. (7) as follows.

\[ \begin{bmatrix} f_A \\ f_B \\ f_C \end{bmatrix} = \begin{bmatrix} x_A & x_B & x_C \\ y_A & y_B & y_C \\ z_A & z_B & z_C \end{bmatrix}^{-1} \begin{bmatrix} x_Q \\ y_Q \\ z_Q \end{bmatrix} \Phi_i. \]  

The coefficient of \( \Phi_i \) is interpolation coefficient and we can finally determine \( c_{ij} \) by Eq. (8).

\[ c_{ij} = \begin{bmatrix} x_A & x_B & x_C \\ y_A & y_B & y_C \\ z_A & z_B & z_C \end{bmatrix}^{-1} \begin{bmatrix} x_Q \\ y_Q \\ z_Q \end{bmatrix}. \]  

Conservation of unsteady force and torque before and after the mapping process are checked and shown in Figure 9 and Table 4. Figure 9 indicates a comparison of summation of \( x \) direction force components on the rotor blade surface between CFD (before mapping process) and FEM (after mapping process) model at each time steps. We can see that unsteady force are sufficiently conserved before and after of the mapping process. Table 4 shows ratios of torque summation on the rotor blade at a certain time step and each torque components, \( x, y, z \), are also sufficiently conserved as well as force.

![Fig. 8 Schematic of the force transfer from a CFD grid point to structural nodes](image-url)
Fig. 9 Conservation check of unsteady force before and after the mapping process

Table 4 Conservation check of torque components before and after the mapping process

<table>
<thead>
<tr>
<th>Torque x (CFD/FEM)</th>
<th>Torque y (CFD/FEM)</th>
<th>Torque z (CFD/FEM)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.000001</td>
<td>0.999999</td>
<td>1.000002</td>
</tr>
</tbody>
</table>

**FREQUENCY RESPONSE ANALYSIS PROCEDURE**

Frequency response analysis is performed by MSC NASTRAN in this study. Some characteristic unsteady nodal force components are extracted by applying FFT procedure to all nodal forces on all structural nodes which received forces from CFD grid points. After the FFT analysis, we can obtain the force amplitude $A$, real part $Re$ and imaginary part $Im$ of a certain frequency component $\omega$ Hz. We then express the dynamic nodal force $P(\omega)$ on structural node $j$ as Eq. (9), which is used in frequency response analysis by varying its frequency $\omega$. Where, $\theta$ is phase angle of the nodal force at structural node $j$ and $\theta$ is calculated by the real and imaginary part as expressed Eq. (10).

$$P_j(\omega) = A_j e^{i(\omega t + \theta_j)}$$  \hspace{1cm} (9)

$$\theta_j = \tan^{-1}(\text{Im}/\text{Re})$$  \hspace{1cm} (10)

Two FEM models, a rotor blade model and a turbine full angular model, are analyzed in this study. Then, the CFD result is coped toward circumferential direction for the turbine full angular model FEM simulation because the CFD analysis is carried out by using periodic boundary condition. These models are composed of quadratic order tetrahedral elements, Tet10. The computational meshes and Tet10 element are shown in Figure 10 and the number of nodes and elements are summarized in Table 5, respectively. Constraints are imposed on the bottom and four sides of the platform in a rotor blade model case and imposed on the inner circumferential surface of the disk in the turbine full angular model case. The material properties are set as Inconel and the detailed values are shown in Table 6. The turbine treated in this study is a blisk and it has small material damping. However, no damping is imposed on the model in this study, which means extremely severe test condition for turbine strength.

Table 5 Number of nodes and elements used in the frequency response analysis

<table>
<thead>
<tr>
<th>Rotor blade model</th>
<th>Turbine full angular model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of nodes</td>
<td>Approx. 25,000</td>
</tr>
<tr>
<td>Number of elements</td>
<td>Approx. 1,340,000</td>
</tr>
</tbody>
</table>

Table 6 Material properties

<table>
<thead>
<tr>
<th>Material</th>
<th>Inconel</th>
<th>Young's modulus</th>
<th>174 [GPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poisson's ratio</td>
<td>0.272</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Density</td>
<td>8.19E-09 [g/cm³]</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**CFD RESULTS AND DISCUSSION**

Time averaged circumferential pressure distributions between the rotor and nozzle and at the exit of the stage are shown in Figure 11. Two-dimensional simulation in Ref. [11] presented that rapid pressure drop occurs at the beginning of the closed nozzle sector between nozzle and rotor blades due to the flow acceleration in the supersonic partial admission turbine. Three-dimensional analysis in this study also indicates similar characteristic. We can also see a pressure increase at nozzle 8~9 pitches between nozzle and rotor blades. Pressure distributions downstream of the rotor blades are weaker than that between nozzle and rotor blades in both the two and three-dimensional analyses. On the other hand, in two-stage simulation of Ref. [11], pressure distributions between 1st stage rotor and 2nd stage stator vary almost sinusoidally. The reason for this almost uniform distributions in this study is probably due to imposing uniform pressure as the outlet boundary condition and this could be a down-side single stage simulation which we should be improved in the future. However, it is interesting to see that pressure distributions qualitatively agree between the two and the three-dimensional analysis in this study. Thus, very useful information on the pressure distribution can be obtained by two-dimensional approach at comparatively reasonable computational cost.

Figure 12 shows instantaneous absolute Mach number contours. High velocity region at the behind of the closed nozzle sector, which spans from the start side to the end side of the closed sector, appears between nozzle and rotor blades in the two-dimensional result, marked (A), as well as in Ref. [10, 11]. On the other hand, 10% and 50% span in the three-dimensional simulation, the working gas stagnates in this region except the start side of the closed sector, marked (B). However, 90% span shows a high velocity region, marked (C), and the distribution behind the closed sector is similar to that of the two-dimensional result. This spanwise distribution can be seen clearly in Figure 13, which shows instantaneous absolute Mach number contours between the nozzle and rotor blades as well as the streamlines. The streamlines coming from hub ~ mid-span source are colored by red and come from mid-span ~ shroud source are colored by blue. Very large separation behind the nozzle closed sector is caused from the hub wall at nozzle one pitch.

Fig. 10 Computational meshes for FEM analysis and Tet10 element
from the start side of the closed sector. Although some working gas which comes from the mid-span ~ shroud source flows into the rotor passages, a part of working gas flows in the circumferential direction near the casing wall toward the end side of the closed nozzle sector. On the other hand, the working gas which comes from the hub ~ mid-span source flows toward the casing at the start side of the closed sector. Then, a part of the gas flows into the rotor passage and the others flow in the circumferential direction near the casing with the working gas coming from the mid-span ~ shroud source. Although the two-dimensional CFD simulation could reproduce a similar trend of the pressure distribution as mentioned above, the flow pattern behind the closed nozzle sector is extremely three dimensional and it cannot be predicted by the two-dimensional analysis.

\[ \Phi = \tau_{ij} \frac{\partial u_i}{\partial x_j} \quad (i, j = 1, 2, 3) \]  (11)

The dissipation area marked (A) in Figure 14 is caused by the passage vortex, shock wave from the adjacent rotor leading edge and tip leakage vortex when the rotor blades passing through open nozzle sector. Thus, the region with high value of dissipation probably occurs in the full admission turbine stages. However, the regions marked (B), (C) and (D) are specific to the partial admission turbine stages. We can see a dissipation area circumferentially spreading from hub to shroud behind the closed nozzle sector, marked (B). This dissipation is caused by the mixing between the circumferential high velocity flow and the stagnation gas existing in the large separation region or the rotor passages. When the rotor blades enter the open nozzle sector, a velocity difference between high velocity flows coming from the nozzle opened sector and stagnation gas behind the closed nozzle sector creates additional dissipations as shown in the region marked (C). Furthermore, rotor blade passages are filled with fluid of high dissipation when the rotor blade enters the closed nozzle sector, marked (D). The source of this dissipation is the separation on the rotor suction surface due to the increase of the incidence and strong shock wave impingement from the adjacent rotor leading edge.

Total-to-total efficiencies are plotted in Figure 15. The values indicated as FullAd are those calculated from the nozzle 12~14 pitches sector in Figure 11, where almost periodic distributions are confirmed, and they seem to be the same values of the full admission turbine in this study. The efficiency drops due to the three dimensional losses are large, 7pt in the full admission (FullAd) and 9pt in the partial admission (ParAd) stage, respectively. Figure 16 also shows the total-to-total efficiency in terms of the ratio of the partial admission to the full admission. In the two-dimensional simulation, the efficiency of the full admission turbine decreases by 7% due to the partial admission. On the other hand, more efficiency decline, 11% of the full admission, occurs in the three-dimensional analysis and this can be attributed to the three-dimensional effect additionally caused by the partial admission configuration.

Fig. 11 Time averaged pressure distributions between inside and exit of the turbine stage, top : between nozzle and rotor blades, bottom : exit of the turbine stage

Fig. 12 Instantaneous absolute Mach number contours

Fig. 13 Instantaneous absolute Mach number contours and streamlines between nozzle and rotor blades

Fig. 14 shows an instantaneous dissipation function contours obtained by the three-dimensional simulation. The dissipation function is described as Eq. (11) and this represents entropy production caused by viscous stresses.

\[ \Phi = \tau_{ij} \frac{\partial u_i}{\partial x_j} \quad (i, j = 1, 2, 3) \]  (11)

The dissipation area marked (A) in Figure 14 is caused by the passage vortex, shock wave from the adjacent rotor leading edge and tip leakage vortex when the rotor blades passing through open nozzle sector. Thus, the region with high value of dissipation probably occurs in the full admission turbine stages. However, the regions marked (B), (C) and (D) are specific to the partial admission turbine stages. We can see a dissipation area circumferentially spreading from hub to shroud behind the closed nozzle sector, marked (B). This dissipation is caused by the mixing between the circumferential high velocity flow and the stagnation gas existing in the large separation region or the rotor passages. When the rotor blades enter the open nozzle sector, a velocity difference between high velocity flows coming from the nozzle opened sector and stagnation gas behind the closed nozzle sector creates additional dissipations as shown in the region marked (C). Furthermore, rotor blade passages are filled with fluid of high dissipation when the rotor blade enters the closed nozzle sector, marked (D). The source of this dissipation is the separation on the rotor suction surface due to the increase of the incidence and strong shock wave impingement from the adjacent rotor leading edge.

Total-to-total efficiencies are plotted in Figure 15. The values indicated as FullAd are those calculated from the nozzle 12~14 pitches sector in Figure 11, where almost periodic distributions are confirmed, and they seem to be the same values of the full admission turbine in this study. The efficiency drops due to the three dimensional losses are large, 7pt in the full admission (FullAd) and 9pt in the partial admission (ParAd) stage, respectively. Figure 16 also shows the total-to-total efficiency in terms of the ratio of the partial admission to the full admission. In the two-dimensional simulation, the efficiency of the full admission turbine decreases by 7% due to the partial admission. On the other hand, more efficiency decline, 11% of the full admission, occurs in the three-dimensional analysis and this can be attributed to the three-dimensional effect additionally caused by the partial admission configuration.
Figure 17 shows the fluctuations of axial torque loading on one rotor blade. The axial torque is normalized by its time-average value. The unsteady force estimations when the rotor blade passes through nozzle 2–9 pitches are obviously different between the two and three-dimensional analysis and the fluctuations appearing in the two-dimensional simulation are larger than that of the three-dimensional analysis. It should be noted that the negative aerodynamic force indicates the force acting against the rotor rotating direction. While the rotor blade experiences some negative forces, its maximum occurs at nozzle 2 pitches in the two-dimensional analysis. However, the negative forces occurring in the three-dimensional analysis are very weak by comparison to that of the two-dimensional result and no negative force appears at nozzle 2 pitches. Static pressure contours and velocity vectors obtained by the two-dimensional simulation is shown in Figure 18. The positions of the rotor blades marked (A) and (B) correspond to nozzle 2 pitches and 4.8 pitches in the Figure 17, where the negative force is exerted. We can clearly see that static pressure of the suction side passages of the two rotor blades, (A) and (B), are larger than that of the pressure side passage. At the suction side passage of the blade (A), a strong detached shock wave occurs at the leading edge of the rotor blade next to the blade (A) and this causes the increase of static pressure. On the other hand, at the suction side passage of the blade (B), a part of the circumferential flow between nozzle and rotor blade slightly inflows the rotor blade passage with extremely positive incidence and this flow behavior causes static pressure increase. Furthermore, at the pressure side passage of the blade (B), a vortex structure appears at the inlet of the passage and this vortex prevents the flow inflowing to the rotor passage. Thus, the static pressure does not increase at the pressure side passage of the blade (B). As just described, the working gas flowing between nozzle and rotor blades behind the closed nozzle sector is one of the sources of the negative aerodynamic force on the rotor blades in the two-dimensional analysis. In contrast, in the three-dimensional simulation, the circumferential flow between nozzle and rotor blades behind the closed nozzle sector is limited to only the casing side as mentioned above. Thus, the influence on the aerodynamic forces on the rotor blades due to the circumferential flow is relatively small in the three-dimensional analysis and the two-dimensional simulation overestimates the fluctuations.
The unsteady aerodynamic force on the rotor blade in the three-dimensional simulation is expressed in the frequency domain by FFT procedure and plotted in Figure 19. Each of the amplitudes is normalized by that of the closed nozzle sector passing frequency component and the frequency is normalized by the rotational frequency of the turbine. The largest component appears at 3 in the normalized frequency, which corresponds to the multiple of 3 to the closed nozzle sector passing frequency. Furthermore, the other components, at the multiples of 9, are also distinguishable. We can see dominant components up to 45 and 45 is the nozzle passing frequency. Other components which may be caused by the existence of the closed nozzle sector appear in a wide region, 15 ~ 42 non-dimensional frequency, and 15, 18 and 21 components have the amplitude on the same level with nozzle passing frequency component. Some frequency components are chosen as the unsteady load condition used in the frequency response analysis described in the next section.

FEM RESULTS AND DISCUSSION

Figure 20 shows the results of the modal analysis of the rotor blade model. The model1 is the first bending mode and the model2 is the first twisting mode. Due to the thickness and the low height of the blade, even the lowest eigenfrequency is very high and it exceeds 30,000 Hz and the eigenfrequency of the model3 exceeds 50,000 Hz. Based on the results of the modal analysis, three different aerodynamic force components, 3 Engine Order (EO), 3EO and 45EO, are chosen and the rotor blade frequency responses to these force components are investigated. Figure 21 is a Campbell diagram of the rotor blade. Although the 3EO component is far away from the eigenfrequencies at the turbine design speed, the component is chosen because it is the largest component as mentioned above. The 36EO component intersects with eigenfrequency curves of the model1 and model2 near the design speed and the 45EO component is chosen because it is the nozzle passing frequency component.

Frequency response of the rotor blade to the 3EO component is presented in Figure 22. The frequency response analysis is conducted by varying the frequency from 2,250Hz (45,000 rpm) to 2,750 Hz (55,000 rpm). The contours (a) represent 3 multiples of the rotor rotation frequency component of the pressure fluctuations on the rotor blade, which corresponds to the distributions of the input nodal force. The displacements (b) are the values at the tip of the leading edge and trailing edge and the Von Mises stresses at the hub of the leading edge and trailing edge are plotted in (c). We can see from the (a) that the force tends to vary approximately over the whole area on the pressure surface. On the suction surface, the force varies in phase out of phase in the vicinity of the shock wave impingement. Although the force component becomes almost the largest, the blade does not respond to the force because the frequencies are enough far from the eigenfrequencies.

Figure 23 shows the frequency response of the rotor blade to the 36EO component. The force frequency is varied from 27,000 Hz (45,000 rpm) to 36,000 Hz (60,000 rpm). There is a phase lag of the force near the mid-span on the pressure surface and large variations appear on the suction surface near and front side of the shock wave impingement position. On the other hand, little force variation occurs at the backside of the suction surface. Both the model1 and the model2 are excited by the force when the force frequency corresponds to these eigenfrequencies. Although the displacements are very small in both modes, approximately 11 MPa of the cyclic stress occurs at the trailing edge hub in the model2 and the response is larger in model2 than model1.

Frequency response to the 45EO component is presented in Figure 24. The simulation is performed by varying its frequency from 30,000 Hz (40,000 rpm) to 40,000 Hz (53,333 rpm). Very complex force variation can be seen on both the pressure surface and the suction surface. A phase lag of the force appears near the mid-span on the pressure surface as well as the 36EO component. Furthermore, the phase of the force near the leading edge of the pressure surface also differs. Large force variations occur around the shock wave impingement on the suction surface and these phases are locally different. Especially, force variation due to the impingement of the shock wave can be clearly observed as spanwise variation. It should be noted that force variation appears not only on the front side of the blade but also near the tip at the backside of the suction surface and this variation can be attributed to the tip leakage flow. The 45EO component also excites both the model1 and the model2 and the displacement is very small in both modes as well as the result of the 36EO component. However, in contrast to the 36EO component case, the model1 is more excited than model2 and approximately 17 MPa of the cyclic stress occurs at the leading edge hub.
The nodal diameter mode shapes and eigenfrequencies of the turbine full anular model are presented in Figure 25 and the Campbell diagram is shown in Figure 26. The 3EO force component does not cross the eigenfrequency curves of the nodal diameter modes near the turbine design speed. However, frequency response analysis is performed in order to investigate the possibility of the each nodal diameter modes excitation by the 3EO aerodynamic force component. Then, the frequency of the dynamic force is varied from 1,000 Hz (20,000 rpm) ~ 3,750 Hz (75,000 rpm). An instantaneous input force distribution is presented in Figure 27 and the distribution tangentially moves. Displacement at the tip of a certain blade leading edge is shown in Figure 28. The 3ND mode shape is excited when the force frequency corresponds to its eigenfrequency but its displacement is very small. Von Mises stress distributions of the 3ND mode shape is also illustrated in Figure 28. Stresses occur not only disk rim but also blade passage, however, the stress value are very small. It should be noted that 1ND or 2ND mode shape are not excited by 3EO force component even if the force frequency corresponds to these eigenfrequencies.

**CONCLUSION**

Three-dimensional unsteady CFD analysis of the supersonic partial admission turbine is conducted in this study. A two-dimensional simulation is also conducted and the approach can qualitatively predict circumferential pressure distributions. However, flow pattern behind the closed nozzle sector indicates strong three-dimensionality, so the flow field expected by the two-dimensional analysis does not agree with three-dimensional result. Large separation occurs from the hub wall behind of the closed nozzle sector and circumferential flow appears only near the casing wall. Three-dimensional spread of the dissipation function caused by the existence of the closed nozzle sector is also observed.
and the rotor passages are filled with the dissipation when the rotor blades enter the closed nozzle sector. The turbine efficiency decreases by comparison with the value of the full admission assumption due to the partial admission configuration in both the two and the three-dimensional simulations. However, the effect of the partial admission configuration to the turbine efficiency is larger in the three-dimensional analysis than in the two-dimensional result. Unsteady aerodynamic force prediction also differs between the two and the three-dimensional approach and the two-dimensional simulation overestimates the fluctuation behind the closed nozzle sector. Turbine vibration response is also investigated in this study by frequency response analysis using FSI procedure. The closed nozzle sector passing frequency component of the unsteady aerodynamic force does not excite the turbine blade because the force frequency is far away from the eigenfrequencies of the rotor blade. On the other hand, 36EO and 45EO force components excite 1st and 2nd mode shapes when the force frequency corresponds to these eigenfrequencies and cyclic stresses occur at the blade hub. Although the mode2 is more excited than the mode1 by the 36EO force component, the 45EO component more excite the mode1 than the mode2. Frequency response analysis of the turbine full annular model indicated that 3ND mode shape is excited by the force of the nozzle closed sector passing frequency component, however, the magnitude of the excitation is very small. Furthermore, 1ND and 2ND mode shape are not excited even if the force frequency corresponds to these eigenfrequencies.
**NOMENCLATURE**

- **M<sub>abs</sub>**: Absolute Mach number
- **P**: Static pressure
- **P<sub>0</sub>**: Total pressure
- **u<sub>i</sub>**: Component of relative velocities
- **η<sub>t-t</sub>**: Total-to-total efficiency
- **τ<sub>ij</sub>**: Viscous stress tensors
- **Φ**: Force at the CFD grid point or dissipation function

**SUBSCRIPTS**

- **in**: Turbine inlet

**REFERENCES**


