Fluid Structure Interaction study in Francis Turbine
Introduction

About QuEST Global
QuEST Global Engineering is a diversified Engineering Services company, employing 2000 professionals across 12 global delivery centers. The company helps customers in Aero engine, Aerospace & Defense, Power Generation, Oil & Gas, and other verticals to cut product development and project costs, shorten lead times, extend capacity and maximize engineering resources availability by providing support across the complete product life cycle from design and modeling through analysis, prototyping, automation, data documentation, instrumentation and controls, embedded systems development, manufacturing support, vendor management, and in-house precision machining. We leverage our local presence and global reach to support engineering cost reduction initiatives for our customers.

About Hitachi
Hitachi, Ltd., (NYSE: HIT / TSE: 6501), headquartered in Tokyo, Japan, is a leading global diversified product & services company with approximately 400,000 employees worldwide. The company offers a wide range of systems, products and services in market sectors including information systems, electronic devices, power and industrial systems, consumer products, materials, logistics and financial services.
Typical Hydropower Plant Setup

Schematic Representation of Hydro Power Plant

- Reservoir
- Dam
- Powerhouse
- Transformer
- Generator
- Hydro Turbine
- Control Gate
- Penstock
- Intake
- Outflow
- Power lines
- Francis Turbine
Basic Concepts

High head turbines are subjected to dynamic forces.

Some significant sources are

1. Interaction between runner and guide
2. Vortex shedding at the trailing edge of vane
3. Cavitation at the trailing edge of the runner vanes

Testing of hydraulic turbine are expensive

Is there any way to reduce the testing cause…?

YES, through CFD…

What is CFD…?

It is the science (art) to replace the partial differential equations (which in this case describe the flow of fluids) with numbers and simple algebraic expressions, and to solve them through iteration in time and/or space to obtain a final numerical description of the total flow field under consideration.

CFD works by dividing the region of interest into a large number of cells or control volumes (the mesh or grid).

It solves the following equations for any problem

- Conservation of Mass
- Conservation of Momentum
- Conservation of Energy
Basic Concepts

Advantages

1. Accuracy
2. Ease-of-Use
3. Speed
4. Powerful Visualizations

Disadvantages or Limitations

1. CFD solutions and accuracy rely upon physical models (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.).
2. Solving equations on a computer invariably introduces numerical errors.
   • Round-off error
   • Truncation error

CFD can be used in:

1. Conceptual studies of new designs
2. Detailed product development
3. Troubleshooting
4. Redesign

CFD analysis complements testing and experimentation. It reduces the total effort required in the laboratory.
Overview

Hydro turbines are desired to work over a wide range of operating conditions due to load variations.

Fatigue:

During dynamic loading, structures fails below its yield strength.

In some cases, when fluid interacts with a solid structure, exerting pressure that may cause deformation in the structure and, thus, alter the flow of the fluid itself.

When the flow induces significant stresses in the solid; however, since the resulting deformation of the solid is small, the flow field is not greatly affected.
1. In the present study, Experimental testing of a Francis Turbine had shown that there is a crack developed at the root of the runner blade after some hours of operation, and this was the probably caused due to the runner blade interaction with the guide vanes.

   When looked at this problem, the flow induces stress in the turbine and the deformation is very small and will not alter flow.

2. In an other study, Experimental testing of a Francis Turbine stay vane had shown that there was vortex shedding from the vane during operation, and this was the probable cause for the observed structural failure in the vane.

   In this case, the vortex shedding frequency when match the resonance of the vane, the vane vibrates violently and will alter the flow.

   These two problems has to solved using computational methods with different approaches. Such problems can be solved using Fluid Structure Interaction (FSI) approach.

   FSI simulations can be broadly categorized as one-way or two-way coupled.

   One way FSI analysis (*Interaction of runner blades with guide vanes*)

   Two ways FSI analysis (*Vortex shedding*)
Case study

1. In Vortex shedding was one of the causes proposed for the failure of the original Tacoma Narrows Bridge (Galloping Gertie) in 1940, but was rejected because the frequency of the vortex shedding did not match that of the bridge. The bridge actually failed by aero elastic flutter.

2. A thrill ride "Vertigo" at Cedar Point in Sandusky, Ohio Suffered the fate of vortex shedding during the winter of 2001, one of the three towers collapsed. The ride was closed for the winter at the time.
1. Executive Summary
   a. Objective
   b. Approach
2. Model and Assumptions
3. Boundary Conditions
4. Results and Discussions
5. Conclusions
Executive Summary

Objective of the study

One Way FSI Analysis
To develop the numerical methodology of Fluid Structure Interaction to find the intensity of stress levels at root of the Francis Runner at operation condition

Two Way FSI Analysis
To develop the numerical methodology of Fluid Structure Interaction to evaluate the ability of an unsteady numerical simulation to accurately reproduce the vortex shedding frequency on the natural frequency of the stay vane.
Executive Summary

Approach

One Way FSI Analysis
1. Perform the unsteady CFD simulation of Francis Turbine and save the result file
2. Map the results (Pressure) on the structure and perform the FEA simulation
3. Check for the maximum stress in the runner

Two Way FSI Analysis
1. 2D Flow field Analysis to Determine Required Mesh Density
2. 3D Modal Analysis of the Vane to Determine Natural Frequencies
3. 3D Flow field Analysis of the Vane to Determine Span wise Variation in Shedding Frequency
4. 2D Flow field Analysis of the Vane to Determine Shedding Frequency at Various Flow rates
5. 3D FSI Analysis at a velocity where the shedding frequency coincides the natural frequency of the stay vane
Model and Assumptions

One Way FSI

✓ A design geometry was used for creating a CFD and FEA model
✓ It should be noted that the design model does not include the fillets
✓ Cavitation effect are neglected do to higher computational time

Geometry

Mesh

Guide vane

Draft tube

RUNNER

Solid region

Fluid Domain

Solid Domain
Two Way FSI

- A design geometry was used for creating a CFD and FEA model
- It should be noted that the design model does not include the fillets
- Cavitation effect are neglected due to higher computational time
Boundary Conditions

Working Fluid: Water

Turbine Material: Structural Steel

One Way FSI
Boundary Conditions

Two Way FSI
Results and Discussions

Observations

One Way FSI Analysis

1. From this analysis, it was seen that the stress were maximum at the root of the runner leading edge.

2. Form the stress harmonic levels is seen at the root that the cause due the interaction of Guide Vane and Runner blade.

3. This analysis doesn't conclude the failure of the runner at root. Fatigue analysis has to be carried our to confirm.

![Graph showing Variation of Normalized Von Mises Stress over Time](image)
Results and Discussions

One Way FSI

The Peak stresses in the runner were predicted to occur at the LE root near the Hub region.

These unsteady pressure effects are quite considerable (~ 4.5% variation) during each vane passing. This results in huge variations in stress levels.

Limitations

Node to node connectivity of fluid and structural mesh will increase the accuracy of the stress prediction.
Results and Discussions

Observations

Two Way FSI Analysis

1. Analysis was carried out with 3 Mesh configurations 13,000 cells, 16,000 cells and 3,50,000 cells. It was seen that 16,000 and 3,50,000 cell had a close match of results and for further analysis 16,000 cells mesh were used.

<table>
<thead>
<tr>
<th>Cases</th>
<th>Mesh Size</th>
<th>First Natural Frequency</th>
<th>Second Natural Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>13,000</td>
<td>91.55 (Hz)</td>
<td>183.1 (Hz)</td>
</tr>
<tr>
<td>2</td>
<td>16,000</td>
<td>85.45 (Hz)</td>
<td>170.9 (Hz)</td>
</tr>
<tr>
<td>3</td>
<td>3,50,000</td>
<td>85.45 (Hz)</td>
<td>164.8 (Hz)</td>
</tr>
</tbody>
</table>

2. It was seen from 3D Modal analysis, the major frequencies (Bending: 73 Hz & Torsional: 155 Hz)
Observations

Two Way FSI Analysis

3. From CFD results, FFT calculations for different points were done and when plotted, it was seen that the frequency had very little effect over the span. This concludes this analysis can be done with 2D to determine the vortex shedding frequency.
Observations

Two Way FSI Analysis

4. For 2D CFD analysis with different inlet velocities,
   a. From the FFT analysis it was seen that at inlet velocity of 12m/s, the vortex shedding frequency is 73 Hz
   b. This concludes that for inlet velocity of 12m/s, the vortex shedding frequency matches the natural frequency of the model
   c. So, FSI analysis can be done with a inlet velocity of 12m/s

<table>
<thead>
<tr>
<th>Velocity (m/s)</th>
<th>Maximum Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>65</td>
</tr>
<tr>
<td>12</td>
<td>73</td>
</tr>
<tr>
<td>13</td>
<td>78</td>
</tr>
</tbody>
</table>
Results and Discussions

Observations

Two Way FSI Analysis

5. The Stress were monitored at the root of the blade. The values were around 10 times less than the yield stress of the material.
Conclusions

One Way FSI

- From this analysis, it was seen that the stresses were developed at the root of the blade which developed as a crack of the turbine due to runner interaction with guide vane.
- Improvement in the results would have been achieved if the there would have been node to node connectivity and using same geometry (Fluid and Surface in common)

Two Way FSI

- Form this analysis, it was seen that the vortex shedding frequency derived from flow induced vibrations match the resonance (bending mode) of the vane. For such lockup conditions, Karman vortices exhibit a strong instability and substantial increase in vibration level.
- The Stress at the root of the blade were around 10 times less then the yield stress of the material. So, cycle calculation would help in determining the fatigue characteristics and number of cycles the vane can sustain.
- Coupled analysis did not show the resonant interaction between fluid and solid that was expected. This is due the oscillations have not reached a constant phase and recommended to run from some more time
THANK YOU