Simulation von Turbomaschinen mit ANSYS CFD 18

Dr. Rolf Reinelt
Agenda

ANSYS TurboSystem
• Workflow & Tools

ANSYS Blade Row solutions
• Methods, Applications & Validation

Aeromechanics
• Flutter & Forced Response

Summary
Design Workflow for a Centrifugal Compressor

- Preliminary design with 1D tools or existing design
- Impeller/rotor geometry
- 2D Throughflow analysis
- 3D Grid generation
- 3D Stage CFD
- Volute geometry + mesh
- 3D Stage + volute CFD

Post-process results (quantitative & qualitative)

Modal analysis

Stress analysis
ANSYS Workbench: Turbo Workflow

Axial Compressor Example

Parametric Geometry

Automatic Meshing

Pre-Stressed Modal Analysis

CFD Simulation
BladeModeler – Meanline Design

• Vista RTD
  – Radial turbine preliminary design

• Vista CCD
  – Centrifugal compressor preliminary design

• Vista AFD
  – Axial fan design
  – Throughflow solver in inverse design or analysis mode
ANSYS TurboGrid

Automated grid generation for bladed turbomachinery components

High quality hexahedral grids

Repeatable

• Minimize mesh influence in design comparison

Scalable

• Maintain quality with mesh refinement

Centrifugal, Mixed-flow & Axial

Parallel (Multithreading)
TurboGrid CAD Support

Improved fidelity & robustness by importing CAD models directly into TurboGrid

- BladeEditor: native blade surfaces & meridional flowpath (Parasolid)
- Direct import: Parasolids (x_b, x_t) & ICEM CFD Tetin (.tin)

Direct import of fillets from CAD

- Exact representation
- High quality mesh & better robustness
ANSYS TurboPre

- Turbo-specific Pre-Processing
- Automated setup
- Multiple component
- Multiple passage
- Interfaces
  - Periodic
  - Rotor/Stator
- Physics
- Boundary Conditions
### ANSYS CFD for Turbomachinery

- **Fast & scalable solver**
- **Low speed to supersonic**
- **Steady/transient**
- **Turbo-specific BC’s**
- **Turbulence & heat transfer**

- **Multiple Frame of Reference**
- **Multi-phase flow**
- **Real fluids**
- **Fluid/structure interaction**
- ...
Multiport MPI

- Single MPI for FLUENT & CFX
- Only 1 CFX executable for all MPI implementations

Solver topology simplifications

- Internal solver topology for 2D mesh regions
- Massive performance improvements for cases with many 2D primitive regions
Turbulence Models

SST Turbulence Model

- Detached Eddy Simulation
- Scale-Adaptive Simulation
- Laminar-turbulent transition
- Streamline curvature & rotation
- 'Automatic' wall functions
- Wall roughness
- Stagnation line flows
Impact of Transition Modelling on Compressor Design

- 4 stage high-speed axial compressor

Transition re-energizes the flow and leads to less separation & lower losses

New 1-equation transition model!
ANSYS Blade Row Analysis Methods

- Steady pitch change
- Transient pitch change
- Transient full-annulus

- Stage interface (mixing plane)
- Frozen rotor interface

- Time domain
  - Profile Transformation
  - Fourier Transformation
  - Time Transformation
- Frequency domain
  - Harmonic Transformation
Turbo Post Processing using Turbo Mode

Turbo plots
- Blade-to-blade (B2B)
- Meridional

Turbo charts
- Blade loading
- Hub to shroud

Turbo report templates
- One component to multi-stage
Live Solution Monitoring – CFX 18 improvements

- Infrastructure and performance
- Support for steady state monitoring
- Support for rotating frames
- Support for batch mode

Cyclic & Polar Plots

- Plotting against period \(\rightarrow\) much better convergence control due to overlay of periods

CFD-Post Highlights 18.0
Performance Maps in CFX

- Convert one or more expressions into Operating Parameters
- Analysis spans operating range – single setup, one execution, ensemble post-processing
- Launched as a single run, results collected into a sub folder
- Designed for massively parallel execution
  - Distribute simultaneous solution of operating points
- Post-processed as a single analysis

1. Define Operating Parameters
2. Use like any expression
3. Specify the operating conditions
4. Launch the run
Robust Design in ANSYS Workbench

Variation Analysis

- Sensitivity
- Calibration of virtual models with physical tests
- Design / data exploration
- Optimization
- Robust design & reliability
Agenda

**ANSYS TurboSystem**
- Workflow & Tools

**ANSYS Blade Row solutions**
- Methods, Applications & Validation

**Aeromechanics**
- Flutter & Forced Response

**Summary**
Blade Row Methods Overview and Objectives

• Blade Row Methods to perform:
  – Aerodynamic, Aeromechanical and Aerothermodynamic analyses

• Methods for both steady-state and transient simulations

• Provide fast & accurate transient blade row solution
  – Using range of pitch-change methods:
    – PT, TT and FT
    – (Full-wheel → Reduced geometry)
  – Harmonic analysis (hybrid frequency/time solution method)
Transient Blade Row Methods With Pitch-Change

Solve on Reduced Geometry

Full Wheel

Reduced Model

Transformation Methods

Profile Transformation (PT)
Small/Moderate Pitch
- Single Stage
- Multistage

Time Transformation (TT)
Small/Moderate Pitch
- Frozen gust
- Single Stage
- Multistage

Fourier Transformation (FT)
Large Pitch
- Frozen gust
- Single Stage
- Multistage
- Blade Flutter

Transient interaction
Correct blade passing frequencies
Harmonic Analysis (Harmonic Balance) and Frequency Based Methods

Turbomachinery flow: Often transient & periodic

- Instead of marching in time to get final steady-periodic state, use HB method to converge fast on steady-periodic state

Assumption:

- Represent solution by Fourier series

\[ \phi(t) = a_0 + \sum_{m=1}^{M} a_m \cos(m \omega t) + b_m \sin(m \omega t) \]

- Number of modes depends on signal complexity

Truly non-linear
Agenda

**ANSYS TurboSystem**
- Workflow & Tools

**ANSYS Blade Row solutions**
- Methods, Applications & Validation

**Aeromechanics**
- Flutter & Forced Response

**Summary**
Blade Flutter & Aerodynamic Damping Calculations

Blade Flutter Analysis:
Determine if the aero loads will damp out blade vibration at natural frequencies

Flutter Margin

Aerodamping in Forced Response analysis

Performance map based on aerodynamic analysis

Performance map based on Aeromechanical analysis

Aerodynamic Damping

Stable

Unstable
Blade Flutter & Aerodynamic Damping

On going V&V of aerodamping under HA method

- All Calculations done with: 1 mode, 15 pseudo-time-step per oscillation cycle
- starting from steady-state solution
- FT-HA (1 mode) about 20x to 30x Faster than FT-Transient (based +100 tspp)
- FT-Transient about 5x to 7x faster than Full-wheel solution

10-to-100X faster
Blade Flutter with Complex mode shapes

Expand blade flutter to more realistic industrial application

Handling of complex mode shape

- Blade/disc instead of blade alone
- Impeller/hub rather than impeller alone
- Tip shroud connectors and snubbers

Enhanced Expand Profile Data tool (β)

- Profile instancing rather than copy & rotate

ND = 4

f = 1759.4 Hz
Agenda

ANSYS TurboSystem
• Workflow & Tools

ANSYS Blade Row solutions
• Methods, Applications & Validation

Aeromechanics
• Flutter & Forced Response

Summary
# ANSYS TurboSystem is the Gold standard

<table>
<thead>
<tr>
<th>Tools for 1D, 2D, 3D</th>
<th>• Vista Tools, Blade Modeler</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Ease of use</strong></td>
<td>• Dedicated tools for pre and postprocessing</td>
</tr>
<tr>
<td><strong>Accuracy</strong></td>
<td>• High quality hexahedral meshes with little effort</td>
</tr>
<tr>
<td></td>
<td>• High solver accuracy</td>
</tr>
<tr>
<td><strong>Speed</strong></td>
<td>• Most advanced Transient Blade Row solutions</td>
</tr>
<tr>
<td></td>
<td>• High Performance Computing (HPC)</td>
</tr>
<tr>
<td><strong>Multiphysics</strong></td>
<td>• Modal and stress analysis</td>
</tr>
<tr>
<td></td>
<td>• Fluid Structure Interaction (FSI) for Flutter analysis</td>
</tr>
<tr>
<td><strong>Optimization and Robust design</strong></td>
<td>• OptiSLang</td>
</tr>
</tbody>
</table>