Turbocharger Design & Analysis Solutions

Bill Holmes
Brad Hutchinson
Detroit, October 2012
Agenda

• ANSYS overview
• ANSYS TurboSystem
• Blade row solutions
  – The ANSYS Transformation methods
• An example: turbocharger compressor analysis
• Summary
ANSYS Vision for Rotating Machinery: Full machine simulation

- High fidelity simulation of all components
- Simulate complex phenomena and processes
  - Unsteady combustion, compressor stall, cavitation, noise, fracture, component interactions, advanced materials.....
- Integrated tool set for all geometry and physics
- Large scale High Performance Computing (HPC) enabled
Turbomachinery @ ANSYS

- Axial and centrifugal compressors
- Axial and radial turbines (Steam & gas)
- Centrifugal, mixed-flow and axial pumps
- Axial and radial fans
- Automotive turbomachinery
- Water turbines
- Wind turbines
ANSYS TurboSystem

- Complete turbomachinery design and analysis in ANSYS Workbench
  - Geometry
  - Throughflow
  - Meshing
  - CFD
  - Thermal
  - Combustion
  - Structural mechanics
  - Rotordynamics
  - Post-processing
  - Optimization

This presentation will focus on ANSYS blade row fluid dynamics for turbocharger compressors
ANSYS Workbench

Parametric Geometry (Meanline & Through-Flow)

Mesh

Robust Design

Analysis
ANSYS Workbench
BladeModeler - Meanline Design

- Vista CCD
  - Centrifugal compressor rotor design
- Real gas capability
BladeModeler – Meanline Design

- Vista RTD
  - Radial turbine preliminary design
ANSYS BladeModeler

- Design comparison
- Visible in meridional sketches, angle/thickness views, blade-to-blade view and 3D view
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 Compressor
ANSYS CFD @ 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
• …
Turbulence Model

- Detached Eddy Simulation
- SST Model
- Scale-Adaptive Simulation
- Wall roughness
- Streamline curvature & rotation
- 'Automatic' wall functions
- Stagnation line flows
- EARS
ANSYS Design Exploration

- Sensitivity analysis
- Design optimization
- Robustness evaluation
Mechanical

- Mechanical deformation
  - Rotational forces
  - Surface pressure loads
- Thermal stress
  - Temperature, Heat flux, ...
- Modal analysis
  - Frequencies
- Blade flutter
  - Aerodynamic damping
- Forced response
  - Transient Rotor-Stator
  - Full 2-way FSI
TBR with pitch change: The ANSYS Transformation methods

- Problem: How to obtain the full-wheel transient solution, but at low cost?

  • Solution: The ANSYS TBR Transformation family of methods
    • New models minimize number of simulated passages, providing enormous efficiency gains and reduced infrastructure requirements
Fast Blade Row Solutions

Steady with Pitch Change

Transient with Pitch Change

Transient Full-Domain

Time Domain

Profile Transformation

Time Transformation

Fourier Transformation

Frequency Domain

Harmonic Transformation

Status:
Release & Beta

Status:
Development
**ANSYS TBR Applications**

**Single-Stage**

**Gust Analysis**

**Turbine**

**Blade Flutter**

**Multi-Stage**

**Gust speed**

**Blade Passage pitch**

**Period**

\[ IBPA = \sigma = \frac{2\pi}{Nb} \cdot j \quad j = 0 \rightarrow (Nb-1) \]

\[ \omega \]

**Damping Coef.**

\[ IBPA \]
Trends @ Turbocharging

- Unsteady-State
  - Rotor-Stator Interaction (Off-Design)
  - Inlet distortion
  - Acoustics
- Turbulent flow with conjugate heat transfer
- Multi-physics
  - Forced response
  - Thermal
- Optimization & Robust Design
- Map Width Enhancement, mixed flow turbine wheels, volute configuration
Multi Physics Modeling

CSM Mesh
3D CAD
Load Transfer
Static & Thermal CSM
CFD Mesh
Aerodynamic CFD
ANSYS Turbo System

Geometry ➔ Mesh ➔ Analysis

Throughflow ➔ Robust Design

ANSYS Workbench
Example Application: Turbocharger Compressor
Turbocharger Compressor Analysis: A Best Practice Example

- Methodology
- Preliminary Design
- Geometry & Meshing
- Impeller-only analysis
- Impeller-diffuser-volute analysis
- Post-processing and interpretation
Methodology

• Pre-CFD
  – Start with geometry that meets design specifications
    • From Vista CCD, CCM, TF and BladeModeler

• Impeller-only analysis
  – The impeller is the heart of the compression system --- understand it first
  – Overall performance: how good can it be, can it be better?
  – Nature of the flow, strengths and weaknesses
  – What factors affect performance? Predictions?

• Whole system
  – Impeller-diffuser-volute analysis
  – Volute-only analysis --- useful?

• Post-processing
  – Quantitative and qualitative
Geometry

• 1-D design developed in VISTA CCD
  – Based on prescribed duty, design constraints

• Impeller Geometry
  – VISTA CCD, CCM → BladeModeler → VISTA TF
  – Make adjustments according to package constraints, design rules, approach etc.
    • Meridional path
    • Blade profile/thickness
    • Hub/backface
    • Tip clearance

• Volute Geometry
  – Spreadsheet based design
    • Mass + angular momentum conservation approach (free vortex)
    • Drives a parameterized DesignModeler geometry
Compressor Design Requirements

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>48 [mm]</td>
</tr>
<tr>
<td>Number of Vanes</td>
<td>6 + 6</td>
</tr>
<tr>
<td>Inlet Temperature</td>
<td>288 [K]</td>
</tr>
<tr>
<td>Inlet Pressure</td>
<td>101.35 [kPa]</td>
</tr>
<tr>
<td>Mass Flow Rate</td>
<td>0.12 [kg/s]</td>
</tr>
<tr>
<td>Pressure Ratio</td>
<td>2.15</td>
</tr>
<tr>
<td>Tip Speed</td>
<td>391 [m/s]</td>
</tr>
<tr>
<td>Shaft Speed</td>
<td>155,733 [rev min^-1]</td>
</tr>
</tbody>
</table>

- High Specific Flow impeller with vaneless diffuser of radius ratio 1.7
- Typical for a gasoline engine with capacity of 1.6L.
- Mid map operating point
Initial Sizing

- Vista CCD used to create a geometry from design requirements
Vista CCD: Output

- Iterate in CCD to achieve acceptable preliminary design
Vista CCM: Input

- Vista CCM used to create a preliminary compressor map
Vista CCM: Output

- Turbocharger compressor is typically operating at off design
Initial Geometry Creation

- VISTA TF requires a geometry: from BladeModeler
  - Push-button solution from Vista CCD

Compressor Shroud Section
Compressor Hub Section
Vista TF: 2D Analysis

• Vista TF is a throughflow (streamline curvature) solver
  – Used to provide further insight into design
  – Contour plots show circumferentially averaged quantities
  – 2D Charts show various design parameters such as loading, incidence and deviation

• Based on results, geometry can be quickly modified and analyzed again
  – Blade and flowpath design improved
  – Can be parametric and optimization can be performed
Vista TF: Qualitative Output

- **Tangential velocity**
- **Solution error**
- **Meridional velocity**
- **Static pressure**
Final Impeller Design

- Final impeller geometry steps prior to meshing
  - Direct to TurboGrid for hex
  - Create fluid flow path for tet meshing
- Volute geometry is generated to match impeller
  - Details later
Final Geometries

Impeller and vaneless diffuser

Volute
Meshing

• **Impeller Mesh**
  - Use a hexahedral mesh: → TurboGrid ATM
  - Pay attention to:
    • Target mesh size
    • Balance
    • Boundary layer resolution
    • Y+
    • Tip clearance
    • Aspect ratio

• **Volute mesh**
  • ANSYS meshing
    - Tets + prisms for boundary layer resolution
    - Local mesh refinement near tongue
    - Match diffuser outlet/volute inlet spanwise mesh distribution
Impeller-only Analysis

- Impeller + part of vaneless diffuser
  - How much of the vaneless space to model?

- Grid refinement study
  - Grid: The biggest factor affecting predictions
  - Tetrahedral Elements Vs. Hexahedral Elements
  - Understand the effect of grid size on prediction
    - Target: “working grid” size with Y+=2
    - Ideally, double/half the grid size in each direction
      - 1/8X, 1X, 8X working grid size
    - Estimate of grid independent-solution
  - Effect of fillets
  - Look at key points on the map
    - Nominal design, near surge line, near choke, choke
Example: Mesh Independence Study

- Impeller + Vaneless Diffuser Analyzed at 155, 733 rpm
- Three operating points
  - Design Flow Rate
  - Near Choke
  - Near Stall/Surge
- Compared Hex mesh vs. Tet mesh
## Mesh Summary

### Hexahedral

<table>
<thead>
<tr>
<th># of Nodes</th>
<th>Blade Y+</th>
<th>Meshing Tool</th>
<th>Meshing Method</th>
<th>Meshing Time</th>
<th>Mesh File Size</th>
<th>Max Vol Ratio</th>
<th>Max Length Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.142m</td>
<td>8</td>
<td>TurboGrid</td>
<td>ATM</td>
<td>1 min</td>
<td>3.67MB</td>
<td>159</td>
<td>2132</td>
</tr>
<tr>
<td>1.12m</td>
<td>4</td>
<td>TurboGrid</td>
<td>ATM</td>
<td>1 min</td>
<td>34.8MB</td>
<td>88</td>
<td>4308</td>
</tr>
<tr>
<td>8.58m</td>
<td>2</td>
<td>TurboGrid</td>
<td>ATM</td>
<td>3 min</td>
<td>273MB</td>
<td>34</td>
<td>3547</td>
</tr>
</tbody>
</table>

### Tetrahedral

<table>
<thead>
<tr>
<th># of Nodes</th>
<th>Blade Y+</th>
<th>Meshing Tool</th>
<th>Meshing Method</th>
<th>Meshing Time</th>
<th>Mesh File Size</th>
<th>Min Angle</th>
<th>Min Quality</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.143m</td>
<td>8</td>
<td>ICEM CFD</td>
<td>Octree</td>
<td>~5 min</td>
<td>56.7MB</td>
<td>0.65</td>
<td>0.01</td>
</tr>
<tr>
<td>1.08m</td>
<td>4</td>
<td>ICEM CFD</td>
<td>Octree</td>
<td>~30 min</td>
<td>601MB</td>
<td>0.31</td>
<td>0.0029</td>
</tr>
<tr>
<td>7.50m</td>
<td>2</td>
<td>ICEM CFD</td>
<td>Octree</td>
<td>~1.5 hr</td>
<td>4.4GB</td>
<td>0.23</td>
<td>1.3e-06</td>
</tr>
</tbody>
</table>
Hex Coarse Mesh
Hex Fine Mesh
Tet Coarse Mesh
Tet Medium Mesh
Tet Fine Mesh
Mesh Independence: Hex vs. Tet Total Pressure

Tet mesh at approximately 8 million nodes still is not as accurate as Hex mesh at 125 thousand nodes!
Mesh Independence: Hex vs. Tet

Isentropic Efficiency

Mesh Sensitivity at Best Efficiency

- Series 1
- Series 2
- Series 3
- Series 4

Number of Nodes

Isentropic Efficiency
Mesh Sensitivity: Hex speedline

Total Pressure vs Mass flow

- Hex coarse
- Hex medium
- Hex fine
- Hex infinitely refined
Mesh Sensitivity: Fine Tet speedline added

Tet mesh at approximately 8 million nodes still is not as accurate as Hex mesh at 125 thousand nodes!
Effect of Fillet

- 1.5 mm fillet included at main and splitter blade root
  - compared to blade geometry without fillet
Fillet Study: Effect on Pressure Ratio

Difference only apparent at/near choke
Fillet Study: Effect on Isentropic Efficiency

Difference only apparent at/near choke
Assembly Analysis

• How much do I really need to model, and using what methods?
  – Impeller-diffuser-volute?
  – Volute only (including part of vaneless diffuser)?
    • Inlet specified from exit of impeller-only analysis
  – Steady state, transient?

• We did the following, for comparison purposes:
  – “Frozen rotor” --- full 360 degrees
  – “Stage” analysis --- one impeller with full volute
  – Transient Rotor Stator --- full 360 degrees
  – Volute only
    • Constant Pt, Tt, flow direction
    • As above but with a spanwise profile
Volute Mesh

- Relatively Coarse Mesh used for Study
- Size: 370,000 nodes
  - Tet Elements = 1.1 million
  - Prism Elements = 0.32 million
- Quality Statistics
  - Average Element Quality = 0.71
  - Min Element Quality = 0.046
Effects of Diffuser and Volute

• Comparison of three different configurations
  – Impeller Only (Single Passage)
  – Impeller + Vaneless Diffuser (Single Passage)
  – Impeller + Vaneless Diffuser + Volute (Full 360, frozen rotor)

• Compare speedlines
  – 155,733 rpm
  – Pt ratio = (Pt outlet/Pt inlet)
  – Isentropic Efficiency
Pressure ratio predictions

155,733 RPM

Mass Flow Rate

Total Pressure Ratio

Impeller Only
Impeller + Diffuser
Impeller + Diffuser + Volute
Isentropic efficiency predictions

155,733 RPM

- Impeller Only
- Impeller + Diffuser
- Impeller + Diffuser + Volute

Isentropic Efficiency vs. Mass Flow Rate
Impeller Behavior

• Now look at **impeller only** performance in two configurations
  1) Impeller-only simulation
  2) Impeller-diffuser-volute simulation

• Speedlines shows Impeller behaves similarly regardless of downstream geometry
  – Pt ratio = \( \frac{P_t \text{ impeller outlet}}{P_t \text{ impeller inlet}} \)
  – Isentropic Efficiency for impeller only

• Significant value in examining individual components to gain insight
Impeller pressure ratio predictions for two configurations

Total Pressure Ratio: Impeller Outlet / Impeller Inlet at 155,733 RPM

- Impeller Only
- Impeller + Diffuser + Volute

Mass Flow Rate
Isentropic efficiency predictions for two configurations

Isentropic Efficiency: Impeller Outlet at 155,733 RPM

- Impeller Only
- Impeller + Diffuser + Volute

Mass Flow Rate

© 2011 ANSYS, Inc. October 17, 2012
Effect of rotating-stationary frame interface type

• Comparison of three interface types between diffuser and volute
  1) Stage (single passage impeller/diffuser, full volute)
  2) Frozen rotor (full 360 degrees)
  3) Transient Rotor Stator

• For all cases
  – Impeller + vaneless diffuser modeled in rotating frame
  – Volute modeled in stationary frame
Effect of interface type on total pressure prediction
Effect of interface type on isentropic efficiency prediction

Full 360 Model with Volute at 155733 rpm

<table>
<thead>
<tr>
<th>Mass Flow Rate (kg/s)</th>
<th>Isentropic Efficiency (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.08</td>
<td>90.0</td>
</tr>
<tr>
<td>0.09</td>
<td>85.0</td>
</tr>
<tr>
<td>0.10</td>
<td>80.0</td>
</tr>
<tr>
<td>0.11</td>
<td>75.0</td>
</tr>
<tr>
<td>0.12</td>
<td>70.0</td>
</tr>
<tr>
<td>0.13</td>
<td>65.0</td>
</tr>
<tr>
<td>0.14</td>
<td>60.0</td>
</tr>
<tr>
<td>0.15</td>
<td>55.0</td>
</tr>
<tr>
<td>0.16</td>
<td>50.0</td>
</tr>
</tbody>
</table>

- Stage
- Frozen Rotor
Post Processing

• Before starting:
  – Make sure solutions are converged!
    • Run with a big enough time step!

• Quantitative
  – Impeller
    • Pt, Tt, Abs. flow angle, isentropic efficiency
    • Distortion factor
    • Blade loading
  – Volute: recovery factor, loss coefficient
  – Estimate grid-independent solution

• Qualitative
  – Blade-to-blade and meridional averaged
  – Unrolled plot at exit of impeller
CFD Results

- Examine results from Compressor Report in CFD Post
CFD Results

- Or use table generation tool in CFD Post to extract custom information at various streamwise locations
Blade Loading Chart near choke
Mass flow = 0.13 kg/s
Relative Mach Number near choke
Mass flow = 0.13 kg/s
Relative Velocity near choke
Mass flow = 0.13 kg/s
Meridional velocity near choke
Mass Flow = 0.13 kg/s
Static pressure near choke
Mass flow = 0.13 kg/s
Relative Mach Number near choke
Mass flow = 0.13 kg/s
ANSYS offers complete turbomachinery design and analysis software

- Geometry
- Throughflow
- Meshing
- CFD
- Thermal
- Combustion
- Structural mechanics
- Rotordynamics
- Post-processing
- Optimization