Automated Design Exploration and Optimization

Clinton Smith, PhD
CAE Support and Training
PADT
April 26, 2012
Agenda

• The path to robust design
• A closer look at what DX offers
• Some examples
The Path to Robust Design

Robust Design is an ANSYS Advantage

Increasing understanding, innovation, ROI

Robust Design

Optimization
- Six Sigma Analysis
- Probabilistic Algorithms
- Adjoint solver methods

Design Exploration
- DOE, Response Surfaces, Correlation, Sensitivity, Unified reporting, etc.

“What if” Study
- Parametric Platform

Multiphysics Solution
- Integration Platform

Single Physics Solution
- Accuracy, robustness, speed...

ANSYS Advantage
Single Physics/Multi Physics

Single physics is the entry point to Simulation

Multi-Physics enables real virtual prototyping

Electromagnetic Force Density with Thermal-Stress and Electromagnetic Force load

2 way coupled with a transient structural solution

Fluid Pressure Distribution

Stress

Deformation of the Stator
“What If?”

Interactively adjust the parameter values and “Update”

Needed for "What If?"

- Parametric CAD Connections
- Pervasive Parameters
- Persistent Updates
- Managed State, Update Mechanisms
- Remote Solve Manager (RSM)
- ...

Parametric Persistence is an ANSYS Advantage!
Design Exploration is an ANSYS Advantage
Optimization
Exhaust manifold design

Six Sigma Analysis

Maximum Displacement should not exceed 1.5 mm

- Input Parameters
- Outlet Diameter of the manifold
- Thickness at inlet
- External Temperature
- Engine RPM

- Response Parameters
- Max Flow Temperature
- Max Deformation
- Max Von-Mises stress

Response Surface showing the effect of engine speed and thickness at outlet on the maximum deformation

All samples report max deformation below 1.5 mm

Uncertainty of input parameters
Integral with Workbench

- Parametric multiphysics modeling with automated updates
- Bi-directional CAD, RSM, scripting, reporting and more...
DesignXplorer is everything under this Parameter bar...

- Low cost & easy to use!
- It drives Workbench
- Improves the ROI!
Design of Experiments

With little more effort than for a single run, you can use DesignXplorer to create a DOE and run many variations.
Correlation Matrix

Understand how your parameters are correlated/influenced by other parameters!
**Sensitivity**

Understand which parameters your design is most sensitive to!
Response Surface

Understand the sensitivities of the output parameters (results) wrt the input parameters.
Goal-Driven Optimization

Use an optimization algorithm or screening to understand tradeoffs or discover optimal design candidates!
Robustness Evaluation

Input parameters have variation!

Make sure your design is robust!

Six Sigma, TQM

Output parameters vary also!

Understand how your performance will vary with your design tolerances?

Predict how many parts will likely fail?

Understand which inputs require the greatest control?
ANSYS DesignXplorerer
Initial vs. Optimized Design

<table>
<thead>
<tr>
<th>Output</th>
<th>Initial Design</th>
<th>Optimized</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_t$ Ratio</td>
<td>1.116</td>
<td>1.126</td>
</tr>
<tr>
<td>$p_t$ Ratio</td>
<td>1.674</td>
<td>1.709</td>
</tr>
<tr>
<td>$\eta$ [%]</td>
<td>71.65</td>
<td>76.25</td>
</tr>
<tr>
<td>Power [MW]</td>
<td>1.208</td>
<td>1.268</td>
</tr>
</tbody>
</table>

Engineers can easily appreciate the value of understanding.
Industry Testimonials

“The ease of using simulation tools has helped to transform our organization from a test-centric culture to an analysis-centric culture.”

Bob Tickel, Director of Analysis at Cummins

“This technology makes it possible to quickly evaluate hundreds of designs in batch processes to explore the complete design space so that we know we have the best possible design.”

Ken Karbon, Staff Engineer, General Motors

“Over the course of the design process, Dyson’s engineers steadily improved the performance of the fan to the point that the final design has an amplification ratio of 15 to one, a 2.5-fold improvement over the six-to-one ratio of the original concept design. The team investigated 200 different design iterations using simulation, which was 10 times the number that would have been possible had physical prototyping been the primary design tool. Physical testing was used to validate the final design, and the results correlated well with the simulation analysis.”

R. Mason, Research, Design and Development Manager, DYSON
Some quick examples
Structural Analysis with Fatigue module

Turbine blade root design

Fatigue life optimization

Input Parameters:
- ds_xtilt
- ds_ytilt
- ds_rootrad

Output Parameter:
- Minimum Life

DOE; Central Composite Difference Algorithm
3 input parameters => 15 design points

Response Surface based on 15 DOE runs

Life Minimum

Optimized Candidate

Goal Driven Optimization using MOGA algorithm

Initial Result
Slit Die

- Need uniform outflow
- Minimize pressure drop
Combustor

- 3 parameters
- Minimize pressure loss
- Minimize mach number

**Sensitivity**
External Aerodynamics

• Volvo XC60 vehicle model
  – Four shape parameters
  – RBF Morph to define shape parameters
  – ANSYS DesignXplorer
    • To drive shape parameters
    • To create DOE
    • To perform Goal Driven Optimization

• Ideal Aerodynamics Optimization Process
  – Capacity
  – Automatic
  – Fast
  – Accurate
Step #1: Baseline Model

- Volume Mesh – TGrid
- Cell Count: 50.2 Million Cells
- Prism Layers: 10 (First Aspect Ratio 10, Growth 1.1)
- Prism Count: 24.4 Million Cells
- Skewness < 0.9
Step #2: CFD Setup

- Boundary Conditions
  - Inlet: Velocity Inlet 100 kmph
  - Outlet: Pressure Outlet, 0 Pa (Gage)
  - Side walls: Wall, no-slip
  - Top wall: Wall, no-slip

- Solver Settings
  - Steady, PBCS, Green Gauss Node Based Gradient
  - Fluid: Incompressible air,
  - Density = 1.225 kg/m³,
  - Turbulence: Realizable K-epsilon,
    Non-equilibrium wall treatment
  - Discretization:
    - Pressure – Standard
    - Momentum, TKE, TDR – 2nd Order

- Solution Controls
  - Courant Number = 200
  - ERF
    Momentum, Pressure = 0.75
  - URFs
    Density = 1.0, Body Forces = 1.0
    TKE, TDR = 0.8
    TR = 1.0
Step #3: RBF-Morph

Note: a separate webinar looks at Fluent MMO in more detail, but this example used RBF-Morph.
Step #3: RBF-Morph

- Fully integrated within FLUENT and Workbench
  - Easy to use
- Parallel => rapidly morph large size models
- Mesh independent solution works with all element types (tetrahedral, hexahedral, polyhedral, etc.)
- Superposition of multiple RBF-solutions makes the FLUENT case truly parametric (only 1 mesh is stored)
  - RBF-solution can also be applied on the CAD
- Precision: exact nodal movement and exact feature preservation.
### Step #4: Setup DesignXplorer

<table>
<thead>
<tr>
<th>ID</th>
<th>Parameter Name</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td><strong>Input Parameters</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>FLUENT (A1)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>P2</td>
<td>boat_tail_angle</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>P3</td>
<td>long_roof_angle</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>P4</td>
<td>green_house</td>
<td>0</td>
</tr>
<tr>
<td>6</td>
<td>P5</td>
<td>front_spoiler</td>
<td>0</td>
</tr>
<tr>
<td>7</td>
<td>*</td>
<td>New input parameter</td>
<td>New name</td>
</tr>
<tr>
<td>8</td>
<td><strong>Output Parameters</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>FLUENT (A1)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>P1</td>
<td>drag_force</td>
<td>N</td>
</tr>
<tr>
<td>11</td>
<td>*</td>
<td>New output parameter</td>
<td>New expression</td>
</tr>
<tr>
<td>12</td>
<td><strong>Charts</strong></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### DOE: Central Composite Design, Face Centered, Enhanced
Results:
# Optimization

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Optimization Study</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Objective</td>
<td>No Objective</td>
<td>Minimize</td>
<td>Maximize</td>
<td>No Objective</td>
<td>Minimize</td>
</tr>
<tr>
<td>3</td>
<td>Target Value</td>
<td></td>
<td>Higher</td>
<td></td>
<td></td>
<td>Higher</td>
</tr>
<tr>
<td>4</td>
<td>Importance</td>
<td>Default</td>
<td></td>
<td>Higher</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>Candidate Points</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>Candidate A</td>
<td>1.4064</td>
<td>-2.1136</td>
<td>0.64871</td>
<td>0.033455</td>
<td>376.26</td>
</tr>
<tr>
<td>8</td>
<td>Candidate B</td>
<td>1.0867</td>
<td>-2.2709</td>
<td>0.68627</td>
<td>2.8351</td>
<td>378.48</td>
</tr>
<tr>
<td>9</td>
<td>Candidate C</td>
<td>1.2998</td>
<td>-2.2249</td>
<td>0.62562</td>
<td>0.93816</td>
<td>377.48</td>
</tr>
</tbody>
</table>
Results:

- Flow Results Discussion
  - Design point 1, 9, 19 & 25
  - Velocity contours
  - Iso-surface of total pressure = 0.0

<table>
<thead>
<tr>
<th>Design Points</th>
<th>Boat Tail Angle (P2)</th>
<th>Long Roof Angle (P3)</th>
<th>Green House (P4)</th>
<th>Front Spoiler Angle (P5)</th>
<th>Drag Force (N) (P1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>388.01</td>
</tr>
<tr>
<td>9</td>
<td>0.000</td>
<td>1.500</td>
<td>0.000</td>
<td>1.900</td>
<td>393.01</td>
</tr>
<tr>
<td>19</td>
<td>1.850</td>
<td>-2.300</td>
<td>-0.700</td>
<td>0.000</td>
<td>372.30</td>
</tr>
<tr>
<td>25</td>
<td>-1.850</td>
<td>1.500</td>
<td>-0.700</td>
<td>0.000</td>
<td>397.33</td>
</tr>
</tbody>
</table>
Design Point #19
Mesh Morphing and Optimizer
Mesh Morpher and Optimizer

You pick your goal (optimization function)

ANSYS FLUENT drives the optimization

The mesh is morphed to the new selected shape (no remeshing)

The process continues until the best shape is found
Fully Integrated Feature—No Added Cost

Fully Integrated Feature in 13.0 and 14.0
Improvements in 14.0

• New improved morphing algorithms
• Ability to use parameters to define objective functions
  – Minimizes the need for UDF writing
• Ability to apply movement/deformation restrictions on specific surfaces
• Ability to plot and track the value of the objective function
• Ability to execute commands before or after each design iteration
Fluent Morpher-Optimization Feature

- Allows users to optimize product design based on shape deformation to achieve design objective
- Based on free-form deformation tool coupled with various optimization methods
Mesh Morphing

Applies a geometric design change directly to the mesh in the solver.

Uses a Bernstein polynomial-based morphing scheme:
- Freeform mesh deformation defined on a matrix of control points leads to a smooth deformation.
- Works on all mesh types (Tet/Prism, CutCell, HexaCore, Polyhedral).

User prescribes the scale and direction of deformations to control points distributed evenly through the rectilinear region.
Process

What if?

Setup Case → Run → Setup Morph → Morph → Evaluate → Choose “best” design

OR

Optimizer

Setup Case → Run → Setup Optimizer → Optimize → Auto → Optimal Solution

Regions
Parameters
Deformation
Optimizer
Example – Simple Sedan

- Define Control Region(s)

Sequential Tabs
Deformation Definition

• Define constraint(s) (if any)

• Select control points and prescribe the relative ranges of motion
Optimization algorithms: Compass, Powell, Rosenbrock, Simplex, Torczon

- Auto
- Objective Function Definition
- Optimization Settings
- Maximum Number of Designs: 500
- Number of Iterations: 1000
- Optimizer Convergence Criteria: 0.0001

- Initialization
- Execute Commands
- Monitor
- Apply
- Optimize

- Optimize!
Results

Incompressible turbulent flow

Objective Function; Minimize Drag

Baseline Design

Optimized Design
Generic Mirror Drag Optimization

Drag Optimization

Initial Value

Final Value

~10%
RBF Morph – An Add-on Module for Mesh Morphing in ANSYS Fluent
Morphing & Smoothing

• A mesh morpher is a tool capable of performing mesh modifications in order to achieve arbitrary shape changes and related volume smoothing *without* changing the mesh topology.
• In general, a morphing operation can introduce a reduction of the mesh quality.
• A good morpher has to minimize this effect, and maximize the possible shape modifications.
• If mesh quality is well preserved, then using the same mesh structure has massive productivity benefits.
The Aim of RBF Morph

The aim of RBF Morph is to perform fast mesh morphing using a mesh-independent approach based on state-of-the-art RBF (Radial Basis Function) techniques.

The use of RBF Morph allows the CFD user to perform shape modifications, compatible with the mesh topology, directly at the solving stage inside the Fluent Solver, by simply adding a single command line in the input file:

```
(rbf-morph '(("sol-1" amp-1) ("sol-2" amp-2)...("sol-n" amp-n)))
```

The final goal is to perform parametric studies of component shapes and positions to find optimal configurations using:

- Design Changes
- Multi-configuration studies
- Sensitivity Studies
- DOE (Design Of Experiments)
- Optimization
RBF Morph Advantages

• Faster and more consistent than remeshing
• Simplifies the parameterization of models
• Cuts out any file i/o associated with traditional remeshing methods
• Fast convergence on new designs from previous results
• Ideal for robust design analysis or optimization
• Truly opens up the realms of design optimisation on *large-scale CFD models* where remeshing and file i/o are very expensive
RBF Morph Features

- The user-friendly RBF Morph addon module is fully integrated within Fluent (GUI, TUI & solving stage) and Workbench
- Mesh-independent RBF fit used for surface mesh morphing and volume mesh smoothing
- Parallel calculation allows to morph large size models (many millions of cells) in a short time
- Management of every kind of mesh element type (tetrahedral, hexahedral, polyhedral, etc.)
- Ability to convert morphed mesh surfaces back into CAD
- Multi fit makes the Fluent case truly parametric (only 1 mesh is stored)
- Precision: exact nodal movement and exact feature preservation.

RBF Morph allows exact prescription of surface movements which opens up several interesting capabilities, including:
- FEM deformed shape (static)
- FSI based on modal FEM analysis
- Target surfaces (STL)
A system of radial functions is used to fit a solution for the mesh movement/morphing, from a list of source points and their prescribed displacements.

Radial Basis Function interpolation is used to derive the displacement at any location in the space.

The RBF problem definition is mesh independent.
How it Works: The Work-Flow

- **RBF Morph execution** requires three steps:
  - Step 1 [SERIAL Fluent] setup and definition of the problem (source points and displacements).
  - Step 2 [SERIAL Fluent] fitting of the RBF system.
  - Step 3 [SERIAL or PARALLEL Fluent] morphing of the surface and volume mesh (available also in the CFD solution stage).
RBF-Morph is Integrated with Fluent
Internal flow example

Here, a pipe is projected onto a previously defined STL shape
Compressor Blade Example

Morphing Preview (A=-10)

Sep 05, 2009
FLUENT 6.3 (3d, pbns, satkw)
External flow example

courtesy of
Ignazio Maria Viola

Morphing Preview (A1=-10, A2=-10)
www.rbf-morph.com

Ship sail rotation
External Flow Example (2)

Sol=sol-01-c, A=0
Surface Grid
Conclusions

• A CFD model can be accurately moulded in a parametric manner within ANSYS Fluent using the RBF Morph Addon Module

• Such parametric CFD model can be easily coupled with automatic optimization tools to steer the solution to an optimal design that can then be exported into the preferred CAD platform (using STEP)

• The proposed approach dramatically reduces the man time required for shape optimisation set-up, thus widening the CFD calculation capability

ANSYS Fluent Adjoint Solver
Summary

Key Ideas
- Fundamentals
- Adjoint equations
- Workflow
- Shape sensitivity
- Gradient algorithm & optimization
- Mesh morphing
- Mesh adaptation

Current Functionality
- Features in Fluent 14 & 14.5

Examples
- Internal flows
- Robust design
- External flows
Key Ideas
Key Ideas - Fundamentals

What does an adjoint solver do?

• An adjoint solver provides specific information about a fluid system that is very difficult to gather otherwise.
• An adjoint solver can be used to compute the derivative of an engineering quantity with respect to all of the inputs for the system.
• For example
  ➢ Derivative of drag with respect to the shape of a vehicle.
  ➢ Derivative of total pressure drop with respect the shape of the flow path.
Key Ideas - Fundamentals

High-level “system” view of a conventional flow solver

**Inputs**
- Boundary mesh
- Interior mesh
- Material properties
- Boundary condition 1
  - Flow angle
  - Inlet velocity
  - ...
- ...

**Outputs**
- Field data
  - Contour plots
  - Vector plots
  - xy-plots
  - Scalar values
    - Lift
    - Drag
    - Total pressure drop
Key Ideas - Fundamentals

HOW ARE CHANGES TO KEY OUTPUTS DEPENDENT ON CHANGES TO THE INPUTS?

Inputs
- Boundary mesh
- Interior mesh
- Material properties
- Boundary condition 1
  - Flow angle
  - Inlet velocity
  - ...
  - ...

Outputs
- Field data
  - Contour plots
  - Vector plots
- xy-plots
- Scalar values
  - Lift
  - Drag
  - Total pressure drop

ADJOINT SOLVER
Fundamentals

- Discrete or continuous adjoint?
- Continuous
  - Mathematically formal.
  - Adjoint is constructed at PDE level.
  - Easier initial implementation.
  - Wall functions, boundary conditions and expansion to richer physics can all be problematic.

- Discrete
  - Numerically formal.
  - Adjoint is constructed at the level of the discretized equations.
  - Mechanical process to construct the adjoint – somewhat challenging.
  - Easier to test.

CHOSEN METHOD
Key Ideas - Workflow

Workflow

• Solve the flow equations and post-process the results as usual.
• Pick an observation that is of engineering interest.
  ➢ Lift, drag, total pressure drop?
• Set up and solve the adjoint problem for this observation
  ➢ Define solution advancement controls
  ➢ Set convergence criteria
  ➢ Initialize
  ➢ Iterate to convergence
• Post-process the adjoint solution to get
  ➢ Shape sensitivity
  ➢ Sensitivity to boundary condition settings
  ➢ Contour & vector plots
Key Ideas

What have we learned so far?

- An adjoint solver can be used to compute the derivative of a chosen observation of engineering interest with respect to all the input data for the system.
- The adjoint equations form a linear system.
- Solving an adjoint problem is not trivial – about as much effort as a flow solution.
- The adjoint solution provides guidance on the optimal adjustment that will improve a system’s performance.
- An adjoint solution can be used to estimate the effect of a change prior to actually making the change.
- Shape sensitivity data can be combined with mesh morphing to guide smooth mesh deformations.
- An adjoint solution can be used as part of a gradient-based optimization algorithm.
- An adjoint solution can be used to guide mesh adaptation.
The adjoint solver is released with all Fluent 14 packages.

Documentation is available
- Theory
- Usage
- Tutorial
- Case study

Training is available

Functionality is activated by Loading the adjoint solver addon module
A new menu item is added at the top level.
Current Functionality

ANSYS-Fluent flow solver has very broad scope

Adjoint is configured to compute solutions based on some assumptions
• Steady, incompressible, laminar flow.
• Steady, incompressible, turbulent flow with standard wall functions.
• First-order discretization in space.
• Frozen turbulence.

The primary flow solution does NOT need to be run with these restrictions
• Strong evidence that these assumptions do not undermine the utility of the adjoint solution data for engineering purposes.

Fully parallelized.

Gradient algorithm for shape modification
• Mesh morphing using control points.

Adjoint-based solution adaption
Current Limitations

• Limitations on models
  • Porous media
  • MRF
  • ....
  • These can be added in time
• Adjoint solver stability
  • For some cases converging the adjoint solver can be difficult
    • Inherently unsteady flow – oscillations in aerodynamic loads can signal that the adjoint may have difficulties.
  • Flows with strong shear of particular character
    • Saddle point, attracting focus, attracting node
  • Stabilization mechanism is in place. Still room for improvement here.
Examples
180° Elbow optimization

Thanks to Hauke Reese
ANSYS Germany
180 Elbow: Optimization Loop

Base design

Final design
External Automotive Aerodynamics - Sedan

Adjoint pressure

Surface map of the drag sensitivity to shape changes
External Automotive Aerodynamics - Sedan
Choose a control volume that encloses the upper part of the rear corner of the vehicle

(Half vehicle)
Baseline drag = 125.8N
Expected change = -1.1N
Actual change = -1.0N

Sequence of exaggerated surface displacement vector fields
External Automotive Aerodynamics - Sedan
Conclusion

Reviewed key parts of adjoint method for CFD

- The origin of the adjoint as a method
- How to interpret adjoint data
- How to use adjoint data in a gradient algorithm
- Combining mesh morphing with the adjoint
- Adjoint-based mesh adaptation

Current Functionality

- Adjoint solver is a full feature in Fluent 14.
- GUI/TUI
- Documentation available
- Training

Examples

- Internal flows
  - Ductwork
  - IC Engine
  - Robust Design

- External automotive flows
  - Drag
  - Downforce in F1