Overview

• Elements of CFD Technology for aeronautical simulations:
  – Grid generation
  – Solver technology
  – Physical modeling (turbulence and transition)
  – Validation simulations
Automated Hexahedra Generation

- **New ICEM Meshing Technology**
  - Generates hexahedral elements near surface
  - Fills the rest with tetrahedral elements
  - Limited user interaction
Unstructured Mesh
Mid Span
Mid Span

Pressure

Wall Shear
• **Requirements for CFD Solver:**
  – Operate at all Mach numbers
  – Robust formulation to handle regions of poor grids
  – Ability to handle low-Re grids ($y+\sim 1$) for high Reynolds number flows
  – Multigrid scalability
  – Scalable parallelization
  – Large problem sizes
CFX Solver: Elements and Volumes

- **Flexibility**
  - Unstructured meshes with most common elements
  - Volumes are mesh-dual of elements

- **Robustness**
  - Pressure based formulation (Rhie and Chow)
  - Coupled solution for mass and momentum
  - Algebraic multigrid

- **Accuracy**
  - Second order and high Resolution schemes
  - Second order in time
Solver: Advection Accuracy

- Bounded 2nd order
- Adaptive $\beta$
- Minimal diffusion
- Robust
- Bounded
- Default

\[
\phi_{ip} = \phi_P + \beta (\nabla \phi)_{ip} \cdot \Delta \vec{x}_{ip}
\]
ISL Space Vehicle Configuration

• Configuration
  – Space vehicle re-entry
  – Courtesy ISL/AFRL AF71 F/G 7311 (Institute Saint Louis)

• Boundary conditions
  – Mach 0.9 to 2.6
  – Re 7.5E6
  – $\alpha = 0$ to $+8.0^\circ$

• Objective
  – Bow shock, separated subsonic flow

© 2006 ANSYS, Inc.
ISL Space Vehicle Configuration

- **Nominal case:**
  - Mach 2.6
  - 3 deg. angle of attack

- **Forces steady in 130 iterations**

- **230 iterations RMS residuals to s.p. round-off**
ISL Space Vehicle Configuration

Normal Force

Moment

- ANSYS, Inc. Proprietary © 2006 ANSYS, Inc.

© 2006 ANSYS, Inc.
Forward Swept Wing

- **Configuration**
  - Forward swept wing
  - TUM (Breitsamter)
  - \( \text{Re} = 0.46 \times 10^6 \)

- **Boundary conditions**
  - \( \text{Mach}=0.118 \)
  - \( \alpha = 0, \ldots, 45 \text{ deg.} \)

- **Grids**
  - 0.15m, 1.2m, 10.0m hex nodes

- **Objective**
  - Low speed external
  - Mesh dependence

Part of this work was supported by research grants from the European Union under the FLOMANIA project G4RD-CT2001-00613.
FSW: Convergence

- 10 million node case
- 8 hours on 26 CPUs (K7 Athlon, 1600 MHz)
- Convergence in ~100 iterations
Turbulence Models

- **One equation models:**
  - Spalart-Allmaras (SA) model
  - KE1E model

- **Two-equation models:**
  - $k-\varepsilon$ model (different variants and extensions)
  - $k-\omega_k$ BSL, SST model
  - Explicit algebraic Reynolds Stress model (EARSM)

- **Second Moment Closure**
  - Launder-Reece Rodi
  - Speziale-Sarkar-Gatski
  - SMC-$\omega$ model

- **Unsteady models**
  - Scale adaptive simulation model (SAS)
  - Detached Eddy Simulation Model (DES)
  - LES
    - Smagorinsky
    - Dynamic (prototype)

- **Innovative wall treatment:**
  - Scaleable wall functions
    - All $\varepsilon$-equation based models
  - Automatic wall treatment
  - All $\omega$-equation based models
**k-ω Automatic Wall Treatment**

- Automatic wall treatment for all ω-equation based turbulence models:
  - k-ω (Wilcox), BSL, SST, SMC-ω.
- Switches gradually between low-Re model and wall function based on normal to the wall grid resolution
- Virtually no dependency of results to y⁺ resolution
  - Green: low-Re mode
  - Red: mixed mode
  - Blue: wall function mode
Comparison of standard and Automatic Wall Treatment

- Flat plate simulation on three different grids with $y+ \sim 2, 10, 80$
- Simulations run with SST model with:
  - Standard low-Re wall treatment
  - CFX automatic wall treatment
- Heat transfer (shown) and wall shear stress (not shown) virtually independent of mesh resolution for automatic wall treatment
- No $y+$ restriction for the user.
Separation Prediction: CS0 Diffuser

- SST model gives improved separation behavior.
- CS0 NASA diffuser testcase (one of the most consistent cases for model validation)
Separation Prediction: NACA 4412 Airfoil

- SST model is designed for aerodynamic flows
- SST Model was developed at NASA Ames
- SST Model was optimized for robustness and generality in CFX (the model developer works for CFX)
CFX Transition Model

- The transition from laminar to turbulent boundary layers is an important physical effect in aeronautics.
- CFX has developed a unique transition model, which can be run inside the code using 2 additional transport equations.
- The model can be applied to complex geometries.
- Helicopter geometry shows transition location at the cabin and the rearward wing system.
- On the cabin, transition is caused by natural transition on the wings due to Bypass transition.
Airfoil with Transition

- Due to change in angle of attack (AOA), the transition location on the upper and lower side takes different positions.

Comparison with experimental data and XFOIL code ($e^n$-method)

Transition location on upper and lower surface as function of AOA

Drag coefficient $C_d$ as function of AOA
McDonnell Douglas 30P-30N 3-Element Flap

Numerous transition locations measured in experiment

Simulations compare well against data

Inlet turbulence intensity specified to match transition location on slat

Re = 9 million, Mach = 0.2, AoA = 8°

Exp. hot film transition location measured as f(x/c)

Main upper transition:
CFX = 0.068  Exp. = 0.057  Error: 1.1 %

Slat transition:
CFX = -0.056  Exp. = -0.057  Error: 0.1 %

Main lower transition:
CFX = 0.587  Exp. = 0.526  Error: 6.1 %

Flap transition:
CFX = 0.909  Exp. = 0.931  Error: 2.2 %
AIAA Drag Prediction Workshop 2003

- Workshop for comparison of CFD codes for simulation of lift and drag of airplane configurations
- Simulation of installation drag of engine nacelle
- Comparison of 18 different contributions mainly from aeronautical research centers and companies.
- Comparison with experimental data for DLR-F6 wing-body and wing-body-pylon-nacelle configuration
Results from all Contributions

- Wide range of results
- Mainly specialized aeronautics codes participation
- Drag prediction is difficult
CFX Results – Drag Polar

- Accurate prediction of lift and drag
- Improved results under grid refinement
Lift Curve Slope WB Case

Lift vs. angle of attack

Workshop Results

CFX Results

© 2006 ANSYS, Inc.
Comparison of Lift Curve Slope

3 different codes

Influence of
• turbulence model
• numerics

SST model proved successful in the workshop
also in other codes
Upper Surface Flow Vis.

Separation Flow

Experimental Oil Flow CFX 5

Overprediction of corner separation zone observed with all turbulence models
Convergence History

- No code converged in the residuals due to oscillation of corner separation bubble
- CFX had lowest iteration count for force convergence of all codes
Forward Swept Wing Airplane

- Complex aeronautical geometry for forward-swept wing.
- Testcase for turbulence model validation
- Experiments TU München (Breitsamter)

Part of this work was supported by research grants from the European Union under the FLOMANIA project
Forward Swept Wing Airplane

- **Configuration**
  - \( \text{Re} = 0.46 \times 10^6 \)
  - \( \text{Ma}=0.118, \)
  - \( \alpha = 0, \ldots, 45 \text{ deg}. \)
  - Experiments TU München (Breitsamter)

- **Grids**
  - ICEM-CFD HEXA
    - Fine: 10 million nodes
    - Medium: 1.2 million nodes
    - Coarse: 0.15 million nodes
Forward Swept Wing: Drag Polar

Drag Polar

Turbulence Model

Grid Resolution

Experiments, Breitsamter
CFX-5, SST model
CFX-5, k-epsilon model
CFX-5, k-omega model

Experiments, Breitsamter
CFX-5, fine grid, SST model
CFX-5, medium grid, SST model
CFX-5, coarse grid, SST model
Forward Swept Wing: Moment Curve

- Moment Curve
- Alpha [degree]
-Moment coefficient

Experiments, Breitsamter
CFX-5, SST model
CFX-5, k-epsilon model
CFX-5, k-omega model

Turbulence Model

CFX-5, fine grid, SST model
CFX-5, medium grid, SST model
CFX-5, coarse grid, SST model

Grid Resolution
Forward Swept Wing: Convergence

- 10 million nodes
- 5h, 26 CPUS, Fujitsu-Siemens/hpcLine, K7 Athlon 1900+, 1600 MHz
- Convergence of residuals and force
Forward Swept Wing: Force Convergence

Lift forces

Run forwardswept 008
Lift force (1st order)

Run forwardswept 001
Lift force (2nd order)
FSI Flutter Testcase

- AGARD 445.6 test case
- Mahogany wood
- $Ma = 0.50 \ldots 1.14$
- Zero angle of attack
FSI Flutter Testcase

- Grids generated with ICEM-CFD HEXA

<table>
<thead>
<tr>
<th>Size</th>
<th>y^+</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>75.286</td>
</tr>
<tr>
<td>Medium</td>
<td>314.033</td>
</tr>
<tr>
<td>Fine</td>
<td>2.419.384</td>
</tr>
</tbody>
</table>
FSI Flutter Testcase

• Eigen-modes from modal analysis
  – Bending mode
  – Torsional mode

<table>
<thead>
<tr>
<th>Mod</th>
<th>Experiment</th>
<th>Simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.59 Hz</td>
<td>9.37 Hz</td>
</tr>
<tr>
<td>2</td>
<td>38.16 Hz</td>
<td>39.07 Hz</td>
</tr>
</tbody>
</table>
FSI Flutter Testcase

- Comparison for flutter frequency for Mach number range 0.50 to 1.14
  - Qualitative agreement with constant offset to experimental data

![Graph showing comparison of flutter frequencies for different grid resolutions and Mach numbers.](image-url)
Summary

• Modern pressure-based CFD methods can handle large and complex aeronautical CFD problems.

• Key technologies:
  – Grid generation
  – Solver (robustness, accuracy, scalability)
  – Turbulence (transition) modeling

• Future
  – More unsteady flows (Scale-Adaptive Simulation)
  – More physical coupling (FSI, acoustics, magneto-hydro, …)
  – Optimization