Rapid Calculations of Three-Dimensional Inlet / Fan Interaction

By Rodrick V. Chima
Presented at the NASA Fundamental Aeronautics 2007 Annual Meeting
Oct. 30 – Nov. 1, 2007
New Orleans, LA

Abstract
Two computational fluid dynamics codes have been merged to permit rapid calculations of inlet / fan interaction. Inlets are modeled using the WIND-US Navier-Stokes code. Fans are modeled using a new three-dimensional Euler code called CSTALL that solves the flow through the entire compression system but models blade rows using body forces for turning and loss. The body force model is described and it is shown how unknown terms in the model can be estimated from other Navier-Stokes solutions of the blade rows run separately. The inlet and fan calculations are run simultaneously and are coupled at an interface plane using a third code called SYNCEX that is described briefly. Results are shown for an axisymmetric nacelle at high angle of attack modeled both as an isolated inlet and coupled to a single stage fan. The isolated inlet calculations are unrealistic after the flow separates but the coupled codes can model large regions of separated flow extending from the lower lip of the nacelle into the fan rotor.
Rapid Calculations of Three-Dimensional Inlet / Fan Interaction

Dr. Rodrick V. Chima
NASA Glenn Research Center
Brook Park, Ohio, USA
Objective

Merge two CFD codes to model inlet / fan interaction including:
• Direct calculation of inlet distortion
• Effects of inlet distortion on fan performance
• Effects of non-uniform fan work on inlet flow
• Fan stall point
• Possibly unsteady inlet buzz or fan rotating stall
• Effect of flow control devices
Standard Practice

Average properties transferred from inlet to fan manually, cannot predict:
- Effects of circumferential distortion on fan
- Effects of non-uniform fan work on inlet
- Unsteady interaction like inlet buzz, compressor stall, etc.
Current Work

Solution variables transferred between codes each iteration

- CSTALL  – steady Euler code gives rapid solutions
- SWIFT  – used to calibrate body forces for CSTALL
- SYNCEX  – couples WIND-US and CSTALL

Inlet modeled with WIND-US
Full 3-D geometry

SYNCEX

Fan modeled with CSTALL
3-D Euler, full annulus
Blade body forces for turning & loss
Previous Work

Hsiao, Dalbey, et al. (Boeing, MIT, 2001)
• Modeled isolated fan stage with WIND
• Body force coefficients calculated from averaged solution and added to WIND using formulation by Gong
• Full 3-D inlet / fan calculated using body forces

Hale, Davis, et al. TEACC code (AEDC, 1998 – present)
• 3-D code used for many military inlet / fan configurations
• WIND used for inlet and spaces between blade rows
• HT0300 streamline curvature code uses correlations for blade row performance
• Currently working with CSTALL as a possible replacement
Current Problem

Technology Development, Inc. (TDI) fan simulator
- 12 in. diameter shrouded fan driven by tip turbine
- IGV, fan, stator, rake array
- Used to pull flow through Lockheed-Martin serpentine inlet
Current Problem

- Serpentine inlet is proprietary and has complicated geometry
- Using axisymmetric nacelle designed by John Abbott, NASA GRC

Computational grid for nacelle
- 272 K points
- 2 zones
Overview of CFD Codes

WIND-US: General 3-D multiblock N-S analysis code (NPARC Alliance)

SWIFT: 3-D multiblock N-S analysis code for turbomachinery (Chima)
  • Central-difference or AUSM⁺ upwind schemes
  • Algebraic or k-ω turbulence models

AVCS: Axisymmetric Viscous Code (Tweedt and Chima)
  • Axisymmetric N-S equations with θ-momentum equation

CSTALL: 3-D unsteady Euler code for compressor stability (Chima)
  • Body forces give turning and loss, calibrated using Swift solutions

SYNCEX: Code for coupling other codes (Tweedt)
  • CFD codes transfer data through SYNCEX using TCP/IP
CSTALL – Numerical Model

Computational model
- 3-D Euler equations
- Steady or unsteady
- Full annulus, periodic wedge segment, or 2-D axisymmetric throughflow

Numerical method
- Central difference or AUSM\(^+\) upwind in space
- 4-stage Runge-Kutta in time

Blade passages
- Axisymmetric Euler equations + body forces + tangential blockage
CSTALL – Numerical Model

Axisymmetric Euler equations
- Cannot predict turning - must specify \( \Delta(rv_\theta) \) or blade angle + deviation (X+\( \delta \))
- Cannot predict loss - must specify entropy rise \( \Delta s \)

Use body forces to produce desired turning and loss
- Body forces can be found analytically using formulation by Frank Marble, 1964
- Forces split into two orthogonal components for turning and loss
- Unknown terms calculated with SWIFT Navier-Stokes code but could be calculated from experimental data or design intent.
Body Force Model - Turning

Turning force normal to relative velocity and span

\[ F_\theta = \rho \frac{v_m}{r} \frac{\partial (rv_\theta)}{\partial m} \]

Rotor angular velocity gradient = f(span) from Swift LE, TE profiles at several operating points
Body Force Model - Deviation

Turning force based on desired flow direction

\[ F_\theta = \rho \frac{v_m}{r} \frac{\partial(rv_\theta)}{\partial m} \]

where

\[ \frac{\partial(rv_\theta)}{\partial m} \approx \left( r_{TE} v_{\theta TE} - r_{LE} v_{\theta LE} \right) \frac{1}{chord} \]

and

\[ v_{\theta TE} = \left[ v_m \tan(\chi + \delta) + r \Omega \right]_{TE} \]

Rotor deviation = \( f(\text{span}) \) from Swift TE profiles at several operating points
Body Force Model - Loss

Loss force parallel and opposed to local relative velocity

\[ |f| = \frac{P}{R} \frac{V_m}{|V'|} \frac{\partial s}{\partial m} \]

Rotor entropy gradient = f(span)
from Swift LE + TE profiles at several operating points
Body Force Model – Interpolation

Body force input profiles are interpolated vs. local corrected flow

\[ \dot{m}_c = \dot{m} \sqrt{\frac{T_0 / T_{0\text{ref}}}{P_0 / P_{0\text{ref}}}} \]

Evaluated independently at each \( \theta \) at blade row LE
Body Force Model – Time Lag

Body forces are lagged to model gradual response to disturbances

\[ \tau \frac{dF}{dt} = F_{steady} - F \]

\[ \tau = \text{Nominal convection time through blade row} \]

Implemented as under relaxation

\[ F^{n+1} = (1 - r)F^n + rF_{steady} \]

\[ r = \frac{dt}{\tau} = O(.01) \]
SYNCEX – c code written by Dr. Dan Tweedt of AP Solutions, Inc.

- Runs in the background and handles data communication, storage, and synchronization between CFD codes
- User routines read and write boundary condition data to SYNCEX
- General interpolation routines provided
SWIFT Calculations

5 block grid, 2.3 M points, 5.3 hr./case on 2 CPUs
Turbine flow modeled as inflow BC

Surface static pressure contours

Comparison of measured and computed total pressure ratios at 100 percent speed
CSTALL 2-D Calculations

Axisymmetric grid, 3,675 points, 25 sec./case on 2 CPUs

Comparison of measured and computed total pressure ratios at 100 percent speed
CSTALL 3-D Calculations

3-D grid, 441 K points, 22 min./case on 2 CPUs
Inflow BC from WIND-US calculation of serpentine inlet

CSTALL calculation of total pressures in the TDI fan with inlet distortion from an S-duct inlet
Coupled Fan / Nozzle

Fan – CSTALL Euler  SYNCEX interface  Nozzle – AVCS viscous

Static pressures in nozzle

lower wall
upper wall
data
Coupled Nacelle / Fan

**Nacelle**
- 272 K points, 2 zones
- WIND-US (full N-S)
- 1.05 hr. on 2 CPUs using PVM

**Fan**
- 132 K points,
- CSTALL (Euler)
- 5.8 min. on 2 CPUs using OpenMP
Coupled Nacelle / Fan

\(M_\infty = 0.25, \alpha = 31^\circ\)

- Isolated nacelle with mass flow exit boundary condition is separated
- Coupled inlet / fan is attached
Coupled Nacelle / Fan

$M_{\infty} = 0.25$, $\alpha = 32^\circ$
• Coupled inlet / fan is separated
• Flow reattaches in the rotor
Coupled Nacelle / Fan

Slight differences in solutions
- Isolated inlet separates 1° earlier than coupled inlet / fan
- Coupled solution shows higher recovery and lower distortion intensity than isolated inlet when separated
Conclusions

CSTALL
• 3-D Euler code gives rapid analysis of entire fan
• Body force formulation gives accurate representation of blade row effects

SYNCEX
• Handles data communication, storage, and synchronization between codes
• Minimal changes to CFD codes

Coupled Inlet / Fan Modeling
• Model problem demonstrated proof of concept
• Minimal interaction in this case

Future Plans
• Complete analysis of serpentine inlet / TDI fan
• Subsonic nacelle / fan
• Supersonic inlet / fan – any suggestions?
References


http://www.grc.nasa.gov/WWW/5810/rvc