Computational Methods Development at Ames

Dochan Kwak
Aeronautical Information Technologies Division
July, 1998

Outline of Talk

- CURRENT THRUSTS
  BACKGROUND / APPROACH
- RECENT ACCOMPLISHMENTS
  HIGHLIGHTS
- CURRENT FOCI
- FUTURE DIRECTIONS
Current Thrusts

- **BACKGROUND**
  Decrease design cycle time  ⇒  Rapid turn-around
  Increase design/process fidelity  ⇒  High accuracy and low variation
  (Accurate flow physics)
  Increase discipline integration  ⇒  Increased range of options via IT

- **OBJECTIVE**
  Develop advanced computational methods to provide appropriate fidelity computational analysis/design capability

- **APPROACH**
  - **Advanced computational methods**
    Enhance / accelerate viscous flow simulation procedures
    Develop hybrid/polyhedral-grid procedure for viscous flow
  - **Methods for evaluation and design process**
    Develop real time transonic flow simulation procedure for production wind tunnel
    Develop intelligent data management technology (IT)
  - **Validation of methods and flow physics study**
Recent Accomplishments / Work In Progress

- ADVANCED METHODS FOR VISCIOUS FLOW SIMULATION

  Enhance / accelerate viscous flow simulation procedures
  • Accelerated convergence of OVERFLOW
  • Developed MPI version of OVERFLOW
  • Study/develop convergence acceleration methods

  Develop hybrid/polyhedral-grid procedure for viscous flow
  • Hybrid-Cartesian grid method in progress
  • High-order polyhedral grid flux calculation formulated

  Develop steady/unsteady incompressible solver with adaptive control

Recent Accomplishments / Work In Progress

- METHODS FOR EVALUATION AND DESIGN PROCESS

  Develop real time transonic flow simulation procedure for production wind tunnel
  • Developed Transonic Overset Potential Solver (TOPS)

  Develop intelligent data management technology (IT)
  • Data compression (feature extraction) method being developed using multiresolution algorithm
Recent Accomplishments / Work In Progress

- VALIDATION OF METHODS AND FLOW PHYSICS STUDY
  - Wingtip and wake vortex study
  - Unsteady incompressible N-S solver and adaptive control
  - Aeroacoustic applications:
    - Flap edge noise and acoustic modeling
    - DNS/LES for jet aeroacoustics

Computational Methods Development

TECHNICAL HIGHLIGHTS
Acceleration of Viscous Flow Simulation

- **FUTURE VISION / PAYOFFS / BARRIERS**
  - Two orders of magnitude improvement in turn around time will enable RANS applications to viscous flow design problems
  - Improved process time will impact design cycle time
  - Existing Navier-Stokes solution procedures are not efficient enough for engineering applications.

- **APPROACH**
  - Turn-around time of current RANS solver can be reduced by advanced convergence acceleration algorithms combined with parallel implementation
  - Advanced methods will be developed to reduce CFD process time from CAD to solution

- **TASKS**
  - Enhancement of OVERFLOW: preconditioning & multigrid, MPI version
  - Development of Cartesian-based hybrid-grid method
  - Development of acceleration algorithms

Acceleration of Viscous Flow Simulations

**Convergence Acceleration for N-S Computations**
NACA4412 M = 0.001 α=0 Re=1.5 Million

- **Convergence Acceleration of OVERFLOW**
  
  **DESCRIPTION**
  Convergence to steady state of the compressible Navier-Stokes code, OVERFLOW, has been accelerated with multigrid and also with low-Mach number preconditioning.

  **ACCOMPLISHMENTS**
  A multigrid algorithm has been added to OVERFLOW. It handles "Chimera" geometries.
  A low Mach number preconditioning option has been added for cases with low free stream Mach numbers (less than 0.2).

  **SIGNIFICANCE**
  Convergence to steady state has been accelerated by a factor of up to 6 for NACA 4412 at M=0.2 and Re=1.5 million case.

  **WORK IN PROGRESS / FUTURE**
  Robustness of the algorithms in multi-zone geometries is being enhanced.

POC: D. Jespersen, 4-6742, jespersen@nas.nasa.gov
T. Pulliam, 4-6417, pulliam@nas.nasa.gov
Acceleration of Viscous Flow Simulations

Effect of Low Mach Number Preconditioning in OVERFLOW

NACA4412 α=0.0 Re=1.5x10^6 Spalart-Almaras

- OVERFLOW / MPI
DESCRIPTION
To reduce turn-around time for obtaining Navier-Stokes Solutions, a parallel version of OVERFLOW has been developed that exploits parallelism at all levels.

ACCOMPLISHMENTS
An MPI version of OVERFLOW has been successfully run on workstations, SP2, T3E, Power Challenge, Origin2000 and Onyx2.

SIGNIFICANCE
Distributed linear solver is at 60-80% of ideal efficiency.
Load balancing allows splitting of large zones across processors and clustering of multiple small zones onto one processor.

WORK IN PROGRESS / FUTURE
The code will be integrated into the standard OVERFLOW and will be a part of the OVERFLOW distribution.

POC: D. Jespersen, 4-6742, jespersen@nas.nasa.gov
T. Pulliam, 4-6417, pulliam@nas.nasa.gov

Acceleration of Viscous Flow Simulations

Illustration of Polyhedral Interface Cells

- Finite Volume Methods for Arbitrary Polyhedral Grids
DESCRIPTION
One key element in developing a hybrid-grid method is how to handle irregular polyhedral cells at interface boundaries. The objective of the current effort is to develop theoretical foundation to achieve very high order accurate solutions for general polyhedral grids.

ACCOMPLISHMENTS
High order flow variables and fluxes for arbitrary polyhedral and polygonal grids have been formulated based on exact integration of polynomials and quadrature approximations.

SIGNIFICANCE
Processing time to generate grids for flows past complex geometries can be significantly reduced by relaxing grid generation requirements, i.e. structured viscous grids on Cartesian outer grids.

FUTURE PLAN
Formulas will be incorporated into existing RANS code to generate polyhedral grid Navier-Stokes capability.

POC: Y. Liu / M. Vinokur, 4-6667, liu@nas.nasa.gov
Acceleration of Viscous Flow Simulations

Structured-Cartesian Hybrid Grid Solution

- Hybrid-Grid Method for CAD to RANS Solutions

**DESCRIPTION**

For 3-D applications: Multiblock grid generation procedure is expensive; Overset grid approach suffers from interpolation inaccuracies; Existing unstructured grid method using triangles needs improvement for viscous flows.

The objective is to develop a hybrid grid method using locally nonoverlapping structured grids on the Cartesian grid background. This requires development of advanced methods for the interface boundaries filled with irregular polygons.

**ACCOMPLISHED**

High order fluxes for irregular polygons have been formulated. Quadratic reconstruction formula has been implemented in 2-D with Roe splitting and an implicit GMRES method.

**SIGNIFICANCE**

The present approach will simplify the grid generation process substantially.

**WORK IN PROGRESS / FUTURE**

High order method for general polyhedral cells will be implemented in 3-D.

POC: M. Delanaye, 4-5176, delanaye@nas.nasa.gov

---

Real Time Transonic Flow Simulation in Wind Tunnel

- **FUTURE VISION / PAYOFFS / BARRIERS**
  - Cost / solution time determines the hierarchy of method selected
  - Quick solutions can be generated in real time to provide critical information for early design/test decision
  - RANS approach is impractical for real time applications during production wind tunnel tests

- **APPROACH**
  - For real time applications, implement fast potential solver (TOPS) using the same grid topology used for OVERFLOW
  - Develop/implement boundary layer correction
  - Develop zonal methods through IT based decision process

- **TASKS**
  - Transonic overset-based potential solver (TOPS)
    Develop TOPS to include wind tunnel wall and support
    Add boundary layer correction into TOPS
    Determine range of applicability and variations
Real Time Flow Simulations

Mach contours: three-zone RAE Wing A with B2 fuselage at M=0.9, α=0

● Transonic Overset Potential Solver (TOPS)

DESCRIPTION
Potential/Boundary-Layer approach for transonic cruise analysis and design can be up to 100 times faster than Navier-Stokes. Full potential capability using existing Chimera zonal grid approach will be valuable for real time application of CFD for wind tunnel tests.

ACCOMPLISHMENTS
Transonic 3-D two-zone wing solver has been completed.
Transonic 3-D three-zone wing/body solver has been completed.
New donor cell search algorithm developed which is up to 100 faster than existing algorithm.

SIGNIFICANCE
Results to date demonstrate viability of full potential Chimera approach, which is compatible with existing OVERFLOW environment.

WORK IN PROGRESS / FUTURE
Complex geometry applications are in progress. The method will be extended to viscous flow, W/T wall interference study and design optimization.

POC: T. Hoist, 4-6032, thol@arc.nasa.gov

Data Management and Feature Extraction Method

● FUTURE VISION / PAYOFFS / BARRIERS
- Intelligent information management is essential to IT based computational simulation methods development
- Method for data compression and reconstruction within specified accuracy can reduce storage requirement and transmission time significantly (i.e. at least an order of magnitude reduction)
- CFD data compression tool (code) is not available

● APPROACH
- Develop multiresolution algorithm to compress 3-D vector quantities and demonstrate the capability using CFD data such as wing tip vortex flow

● TASKS
- Develop data compression and feature extraction algorithm / code
Data Management and Feature Extraction Methods

Compression of Analytic Data Using Multiresolution Algorithm

DESCRIPTION
Numerical solution is, in general, only an approximation to the physical solution therefore for many purposes truncation of numerical data is acceptable. An efficient data compression algorithm for large 3-D data sets is desired for storage and transmission of data.

ACCOMPLISHMENTS
A prototype 3-D truncation compression algorithm has been programmed in Matlab and Fortran. This code is based on Harten's multiresolution algorithm with smooth basis functions.

SIGNIFICANCE
Since most numerical flow solutions are smooth almost everywhere, this algorithm should provide efficient CFD data compression. Truncation even with high accuracy (e.g. 99%) may allow significant compression thus reducing storage and transmission requirements.

WORK IN PROGRESS / FUTURE
Will develop a 3-D CFD data compression code.

CFD Application to Safety

Characterization of Wake Vortex

DESCRIPTION
To develop predictive capability for aircraft wake, it is important to characterize the initial tip vortex formation/roll-up process and the near wake behavior. The objective of the present study is to assess current capabilities and define requirements for numerical simulations.

ACCOMPLISHED
Tip vortex formation and roll-up process have been studied in detail addressing the roles of numerics and turbulence model in prediction. Applicability of a simplified Navier-Stokes solver using velocity-vorticity formulation was investigated for wake modeling. This model was compared with 0.03 scale model test of B747 (60x120 data by Rossow et al.).

SIGNIFICANCE
The current work is an attempt to develop a physics-based wake vortex prediction model.

WORK IN PROGRESS / FUTURE
Grid adaptation methods are being investigated to reduce the grid density requirement.

POC: J. Dacles-Mariani, 4-5369, dacles@nas.nasa.gov
Wake of a Lifting Wing

Initial Vortex Formation
- less than 5 wingspans behind wing
- tip vortex starts its roll-up process from wing

Near-Wake
- 5-6 wingspans behind wing
- roll-up effectively completed

Intermediate-Wake
- 6-7 to a few hundred wingspans behind wing
- vortex well established

Far-Wake
- greater than a few hundred wingspans behind wing
- diffusion across the vortex takes place

COMPUTED RESULTS
Test Conditions for B-747 Modeled in the 80 x 120 ft Wind Tunnel*:

Freestream Velocity = 131 ft/sec
Dynamic Pressure = 20 lb/ft²
Distances = 81 and 162 ft

<table>
<thead>
<tr>
<th>Parameter</th>
<th>B-747</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scale of Model</td>
<td>0.03</td>
</tr>
<tr>
<td>b (wingspan)</td>
<td>70.5 in.</td>
</tr>
<tr>
<td>c (wing chordlength)</td>
<td>10.1 in.</td>
</tr>
<tr>
<td>Re</td>
<td>660,000</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>6.96</td>
</tr>
<tr>
<td>S (wing planform area)</td>
<td>4.94 ft²</td>
</tr>
</tbody>
</table>

*Rossow, et al. (1993)
WINGTIP VORTEX VALIDATION STUDY
INS3D-LrPCODE. NACA0012. Re = 4.6e6, α = 10°. GRID SIZE = 115x157x83

32 in. x 48 in. FML Wind Tunnel Velocity Magnitude Comparison between Exp. and Computation

Computed/Measured Domain

Inflow Boundary Outflow Boundary

WINGTIP VORTEX STUDY
Turbulence Model/Grid Density Assessment

Experiment

1.5 million grid points (SA model)

1.5 million grid points (BB model)

2.5 million grid points (SA model)
COMPUTED RESULTS
FULL NAVIER-STOKES FORMULATION

Computed w-velocity progression in the streamwise direction

Comparison of the up- and downwash velocity profile with measured data at x/c = 162 ft

AXIAL PROGRESSION OF WINGTIP VORTEX IN NEAR-WAKE

NACA 0012 WING WITH ROUND TIP
Re = 4.6 Million
$\alpha = 10^\circ$
Incompressible Navier-Stokes Methods

Axial progression of wingtip vortex in near-wake NACA 0012 wing with round tip: Re=4.6x10^5, α=0°

- Fractional-Step Method for Time-Accurate Incompressible Navier-Stokes Computations

DESCRIPTION

Both artificial compressibility method and preconditioned compressible N-S methods require subiterations for time-accurate solutions of incompressible Navier-Stokes equations.

The objective is to develop an efficient unsteady incompressible method for 3-D applications using fractional step method.

ACCOMPLISHMENTS

Implicit Runge-Kutta scheme of order N and GMRES have been implemented.

Spalart-Allmaras as well as Baldwin-Barth model have been implemented.

Validation cases include Pulsatile flow through constricted channel and a wake vortex problem.

SIGNIFICANCE

The CPU time requirement for unsteady calculations has been reduced by a factor of 3-5.

WORK IN PROGRESS / FUTURE

Robustness at high Reynolds number with highly stretched grid cases is being enhanced.

Exploratory Research - Noise Correlation

Far-Field Noise from A Family of Single Element NACA Wings

- Correlation of Wingtip Noise with Mean Flow Parameters

DESCRIPTION

Correlations between steady, mean-flow parameters and radiated farfield noise were found for a single-element wing with a round tip.

APPROACH

Steady, incompressible RANS (INS3D-UP) calculations provided detailed flow information for the wing investigated. An experimental test performed at LaRC (Brooks and Marcolini, 1986) provided farfield noise measurements. The two investigations were combined to generate a correlation that held for combinations of aspect ratio, Reynolds number, and angle of attack.

SIGNIFICANCE

Correlations help understand what flow features produce farfield noise, suggest ways to reduce noise, and make possible an acoustic model that would use steady flow parameters.

WORK IN PROGRESS / FUTURE

The same correlations are being applied to the flap side edge of a multi-element wing.

POC: D. Mathias, 4-0836, mathias@nas.nasa.gov
11 INCH SSME HPFTP IMPELLER

Pressure

Hub surface colored by static pressure

Shrouded impeller
6 full blades
6 long partials
12 short partials

Impeller shaft speed: 6322 rpm
Impeller exit wheel speed:
3641 in/sec
Impeller exit diameter: 11 inches
Reynold Number: 1.81e+5
Reference length: 1 inch
Reference velocity: 284 in/sec
Fluid medium: water at 70 F

SSME-HPFTP Impeller
downstream of impeller exit plane
(51% of blade height)

Impeller exit radius:
5.5 in.

V* : Impeller exit wheel speed (303.5 ft/sec)

R = 5.57 in.
R = 5.833 in.

Flow angle (deg)
Circumferential angle from suction side (deg)

---

Experiment
Computation
Exploratory Research-CFD Application to Biofluid

- CFD-Assisted Development of a Long Term Ventricular Assist Device (VAD)

**DESCRIPTION**

Short-term VAD (two-week pump) has been successfully developed with the aid of INS3D. For long term use, the device has to be controllable adjusting to the patients activity which requires full unsteady flow analysis.

The objective is to develop a long term VAD by implementing adaptive control in conjunction with unsteady flow simulations.

**ACCOMPLISHMENTS**

Developed unsteady flow simulation procedure which can be used for the entire configuration.

Pulsatile flow through various inlet cannula designs were compared.

**SIGNIFICANCE**

Development of the device could not have been possible without CFD capability.

**WORK IN PROGRESS / FUTURE**

Unsteady flow simulation for the entire geometry including an adaptive control model will be attempted.

POC: C. Kiris, 4-4485, kiris@nas.nasa.gov
D. Kwak, 4-6743, dkwak@mail.arc.nasa.gov
Historical Examples

• MAJOR ACCOMPLISHMENTS

Ames has produced the original basic advances in CFD necessary for aeronautic applications and has helped extend CFD into many new and critical aerospace applications.

Following are only a few examples in critical CFD processes illustrating Ames contribution in the recent past:

Grid Generation advances and codes:
Developed elliptic, hyperbolic, unstructured point-insertion and local optimization, patched and overset (Chimera) domain decomposition grid procedures.
Resulting codes include
3D GRAPE, HYPGEN, DCF, SAGE, SURGRID, PRISIM, DELAUNAY

CFD Algorithm advances:
Developed following algorithms and disseminated throughout the US:
- MacCormack explicit algorithm
- Beam-Warming approximate factorization algorithm
- AF1/AF2 algorithm for small disturbance and potential flow
- Pulliam-Chaussee diagonalized ADI algorithm
- Steger-Warming flux-splitting algorithm
- Pseudo compressibility algorithm for incompressible Navier-Stokes equations
- Linear-reconstruction unstructured scheme

Flow Solvers and codes:
Small-disturbance, potential and BL: Bailey-Ballhaus, LTRAN2, TAIR/TWING, BL3D
Euler/Navier-Stokes: ARC2D/3D, TNS, PNS, TIGER, OVERFLOW, CNS
Incompressible: INS3D family of codes
Aero-elastic: ENSAERO
Rotorcraft: TURNS, OVERFLOW_Rtrcrft
Turbomachinery: ROTOR, STAGE
Historical Examples

• MAJOR ACCOMPLISHMENTS
  Ames has produced the original basic advances in CFD necessary for aeronautical applications and has helped extend CFD into many new and critical aerospace applications.

Following are only a few examples in critical CFD processes illustrating Ames contribution in the recent past:

Grid Generation advances and codes:
Developed elliptic, hyperbolic, unstructured point-insertion and local optimization, patched and overset (Chimera) domain decomposition grid procedures.
Resulting codes include
3D GRAPE, HYPGEN, DCF, SAGE, SURGRID, PRISIM, DELAUNAY

CFD Algorithm advances:
Developed following algorithms and disseminated throughout the US:
  MacCormack explicit algorithm
  Beam-Warming approximate factorization algorithm
  AF1/AF2 algorithm for small disturbance and potential flow
  Pulliam-Chaussee diagonalized ADI algorithm
  Steger-Warming flux-splitting algorithm
  Pseudo compressibility algorithm for Incompressible Navier-Stokes equations
  Linear-reconstruction unstructured scheme

Flow Solvers and codes:
Small-disturbance, potential and BL: Bailey-Ballhaus, LTRAN2, TAIR/ TWING, BL3D
Euler/Navier-Stokes: ARC2D/3D, TNS, PNS, TIGER, OVERFLOW, CNS
Incompressible: INS3D family of codes
Aero-elastic: ENSAERO
Rotorcraft: TURNS, OVERFLOW, Rtrcrft
Turbomachinery: ROTOR, STAGE
Historical Examples

- **MAJOR APPLICATIONS**

  **Aerospace:**
  - Coupled aerodynamic-structured prediction of flutter
  - Full fighter aircraft performance predictions: transonic cruise (F-16), high alpha (F-18 with Dryden), STOVL (Harrier)
  - Full rotorcraft performance prediction (e.g. V-22)
  - Full simulation of the Space Shuttle stack performance (with JSC)
  - "Tip-to-tail" hypersonic aircraft performance prediction
  - Single- and multi-stage turbomachinery performance prediction
  - Simulation/redesign of the SSME Hot Gas Manifold
  - Coupled aero-optics-acoustics analysis of a space observation system, SOFIA

  **Non-aerospace: Examples of technology reapplications**
  - First simulation of Artificial Heart (Penn State)
  - Development of the NASA/DeBakey Ventricular Assist Device
  - Simulation of naval vehicles (submarine and propeller with the navy)

Future Directions

- **RESEARCH**

  **Algorithm:**
  - Convergence Acceleration
  - Viscous Separated / Unsteady Flow Methods
  - Solution Adaptive Method
  - Parallel Computing and Real-Time CFD
  - High-speed and chemically reacting flow
  - LES/DNS (Leveraged by CTR's Advances)

  **Flow Physics:**
  - Transition
  - Turbulence
  - Combustion/ Real gas effects
  - Aeroacoustics

  **IT:**
  - Computational Tool for Non-experts (User-Friendly)

  **Software:**
  - Outsource (Commercial Vendors)