1995122307 [redacted] N95-28728
404927
12p

APPLICATION OF CFD IN AERONAUTICS AT NASA AMES RESEARCH CENTER

Catherine M. Maksymiuk, Francis Y. Enomoto,
and William R. Van Dalsem
NASA Ames Research Center
Moffett Field, California

Summary

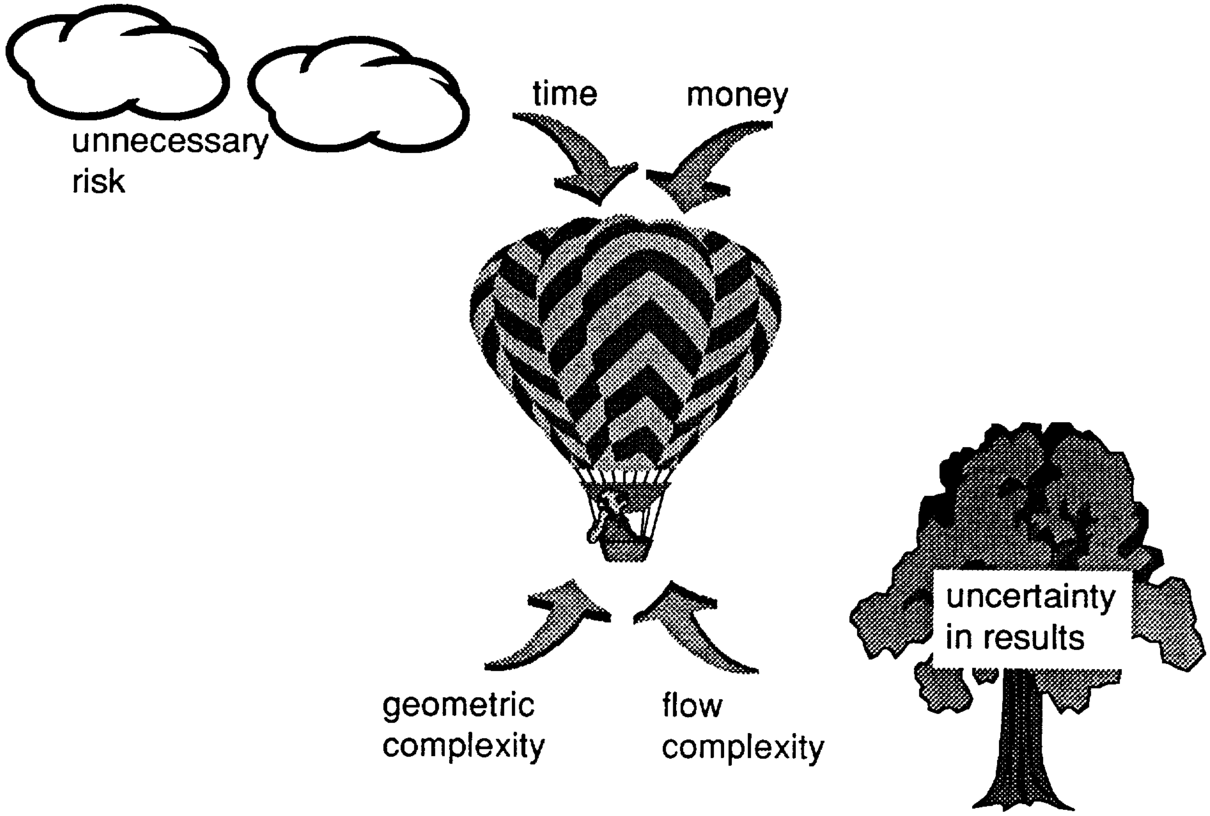
The role of Computational Fluid Dynamics (CFD) at Ames Research Center has expanded to address a broad range of aeronautical problems, including wind tunnel support, flight test support, design, and analysis. Balancing the requirements of each new problem against the available resources - software, hardware, time, and expertise - is critical to the effective use of CFD. Several case studies of recent applications highlight the depth of CFD capability at Ames, the tradeoffs involved in various approaches, and lessons learned in the use of CFD as an engineering tool.

Outline

- **CFD as an Engineering Tool**
- **CFD Process in Action--4 Case Studies**
- **Assessment of Current Technology**
- **Conclusions**

[redacted]

Balancing Requirements and Resources



When CFD is used as an engineering tool, care must be taken to achieve an appropriate balance between a project's modeling requirements and the associated cost. Time and cost constraints become direct limitations on how much complexity is retained in the surface model and how much approximation is made in the fluids model. If the models are oversimplified, the resulting "cheaper, faster" process might produce results that are not useful. Similarly, a simulation which models every imaginable detail might take so long that the results, however accurate, are too late to influence design decisions. The CFD engineer must keep his resources and requirements in equilibrium and must still avoid undue risk and uncertainty. This thought process should accompany every CFD application.

Applied CFD at NASA-Ames

- Wind Tunnel Support
  - Model/Support/Tunnel Design
  - Wall/Sting Interference Prediction
  - Tunnel Data to Flight Extrapolation
  - Tunnel Data Analysis
- Flight Test Support
  - Flight Test Design
  - Flight Data Analysis
- Vehicle (or Component) Design & Analysis:
  - Concept Downselect & Preliminary Design
  - Design Optimization
  - Detailed Analysis

CFD is applied in many ways at Ames. These projects range from short-term, limited-scope analyses to detailed design studies or intensive analyses lasting many months. A corresponding range of tools and expertise are available to address these problems. The following slides will highlight some recent applications projects, showing how we rely on CFD and noting some continuing problems in the process.

Advanced Subsonic Transport

M. Aftosmis, J. Melton

Problem

Investigate aft flow environment

Constraints

Turn-around ASAP

Approach

Vortex-dominated flow required high degree of geometric fidelity
Turn-around time precluded Navier-Stokes or any time-intensive grid system
Cartesian Euler with solution adaption offered quickest gridding

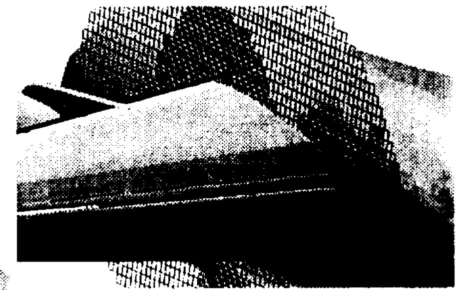
Outcome

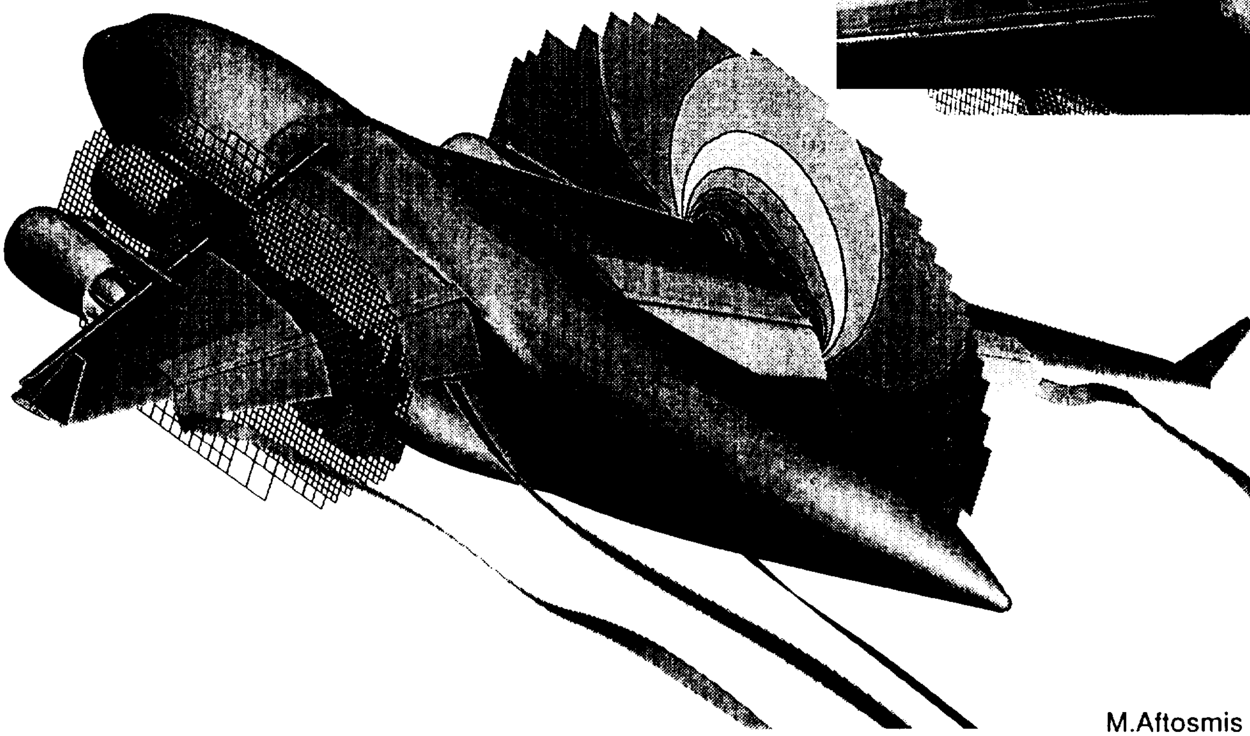
Dominant flow structures were identified and results were coordinated with lower-order methods and water tunnel experiments.
Although the strength of the vortex structures is questionable due to the lack of viscosity, the trajectories are thought to be accurate.
The requesting agency, flight test center, and aircraft manufacturer were favorably impressed with the results and their timeliness.

Lessons Learned

CAD and surface grid generation are still a labor-intensive bottleneck.

Planar cut, Isobars, and Streamribbons





M.Aftosmis

Navier-Stokes Analysis of Flow about a Flap Edge

D. Mathias, K. Roth, J. Ross, S. Rogers, R. Cummings

Problem

Increased interest in aerodynamics and acoustics of high lift systems prompted an intensive analysis using CFD and wind tunnel tests

Constraints

Accurate representation of physics needed for full understanding of flow

Approach

Moderately complicated geometry was readily handled with algebraic and hyperbolic overset grids; a grid scripting system was created to facilitate handling future model with similar topology.

Flow conditions required the influence of viscosity but compressibility could be neglected.

Outcome

Initial results were used to locate pressure taps on model and proved to be an excellent prediction of wind tunnel data.

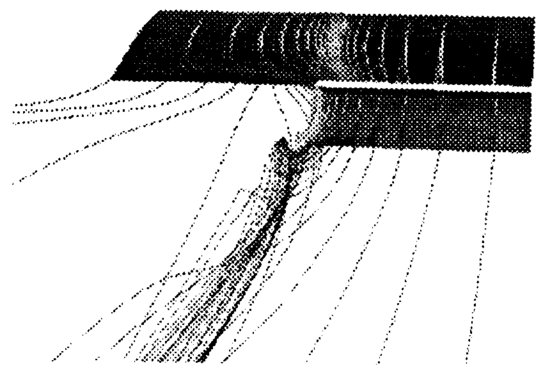
Lessons Learned

Engineering an overset grid system for a class of problems requires a significant investment up front, but can pay off in later similar problems.

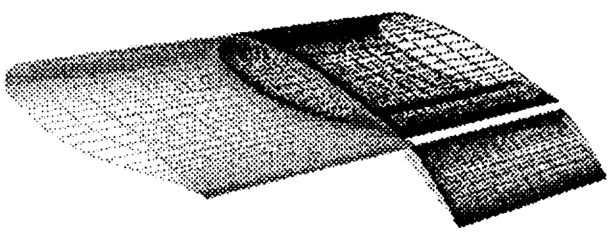
Proper location of overset grid boundaries is critical to obtaining desired convergence.

Incompressible Navier-Stokes Analysis of a Half-Span Flap

streamlines colored by static pressure



overset surface grids



Optimization of a Transonic Business Jet Wing

J. Gallman, M. Madson, R. Kennelly, R. Chandrasekharan, V. Hawke, M. Hinson, D. Saunders, R. Mendoza, J. Reuther

Problem

Design a new transonic business jet wing using numerical optimization and CFD

Constraints

Limitations on wing geometry due to manufacturing considerations

Performance requirements for $C_{D_{WAVE}}$, $C_{L_{MAX}}$ and no shock-induced separation

Short design period requiring efficient CFD and geometry modeling tools

Approach

Set of CFD codes included wing-alone full potential with numerical optimization, full configuration Cartesian full potential, wing-alone full potential with strip boundary layer, and wing-body structured Euler

Common geometry input from an automatic surface geometry program

Outcome

Strong influence of fuselage and nacelle on wing design

Design met most goals and was experimentally validated

Lessons Learned

Future aerodynamic design tools need to include fuselage and nacelle modeling, more accurate maximum lift estimates, and boundary layer effects

Oblique All-Wing Supersonic Transport

R. Kennelly, D. Saunders, S. Cheung, C. Lee

Problem

Aerodynamic design of an Oblique All-Wing supersonic transport

Constraints

Thickness constraints due to passenger cabin requirements
Pitching and rolling moment constraints

Approach

Set of CFD codes included airfoil and wing full potential with numerical optimization, airfoil full potential with boundary layer, full configuration Cartesian full potential, and overset Navier-Stokes methods
NURBS representation used for design variables in airfoil aerodynamic optimization
Chord and thickness tapers designed with numerical optimization

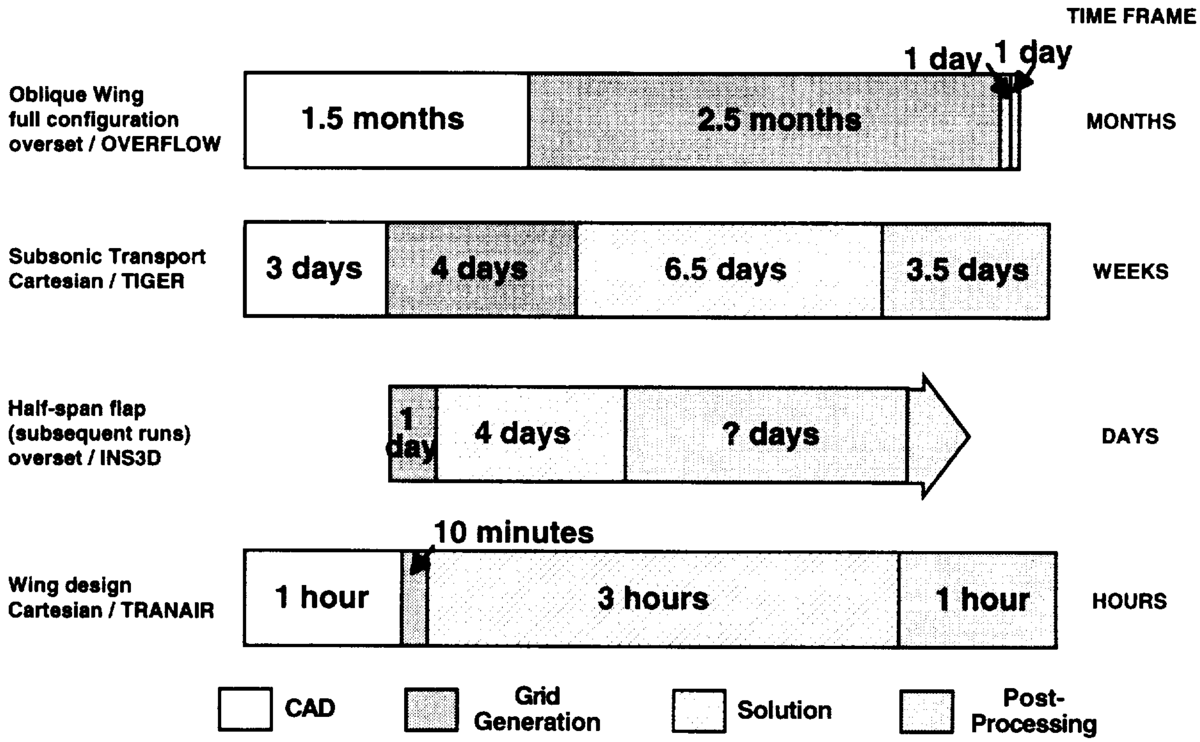
Outcome

Extensive geometric parameter studies led to an asymmetric planform design
Design met goals and was experimentally validated

Lessons Learned

Parametric surface definition was useful for making numerous design iterations
Full configuration analyses were indispensable in design process, but hampered by CAD to surface grid bottleneck
Navier-Stokes analyses took months

Process Time for Case Studies



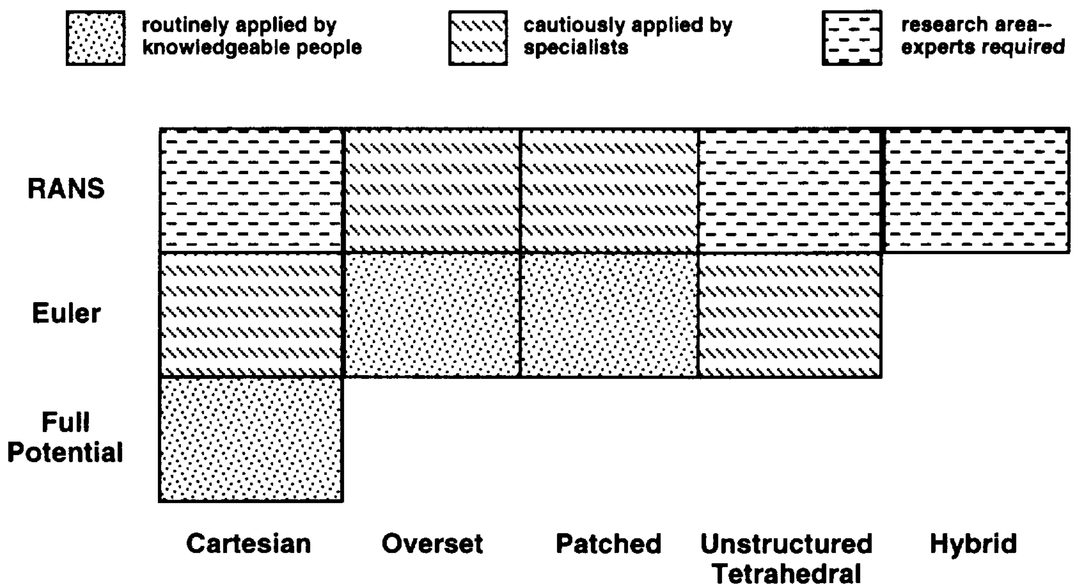
It is instructive to compare process times for the various methods and applications. The design cycles for the Oblique Wing and the Business Jet Wing took 1 year and 4 months respectively; the full processes are not represented here. The Navier-Stokes analysis of the Oblique Wing took several months. The full configuration was a complicated CAD model, and the grid required considerable effort. Once a complete overset grid system was obtained, changing the configuration by moving or removing any of the modular parts was relatively simple, and several such configurations were studied. Post-processing was automated to extract desired information from the solution files.

The full-configuration transport analysis is an example of a short-term, limited-scope project. The short process time was a major goal, which dictated the choice of method. The elapsed time for CAD modeling is a little deceptive, since several people worked on the model.

The Half-span Flap analysis is similar to the Oblique Wing design study. Considerable time was invested to set up a system that could be used for future work on similar configurations; such studies can now be run in a few days. Because the resulting database is used for continuing study, especially in connection with wind tunnel data, the post-processing time is shown as indeterminate.

The most efficient process shown is a TRANAIR run, concluded in only a few hours. The CAD process was actually a surface modeling process not involving conventional CAD software. Analytical surface modeling was done specifically to avoid having to work with a CAD model.

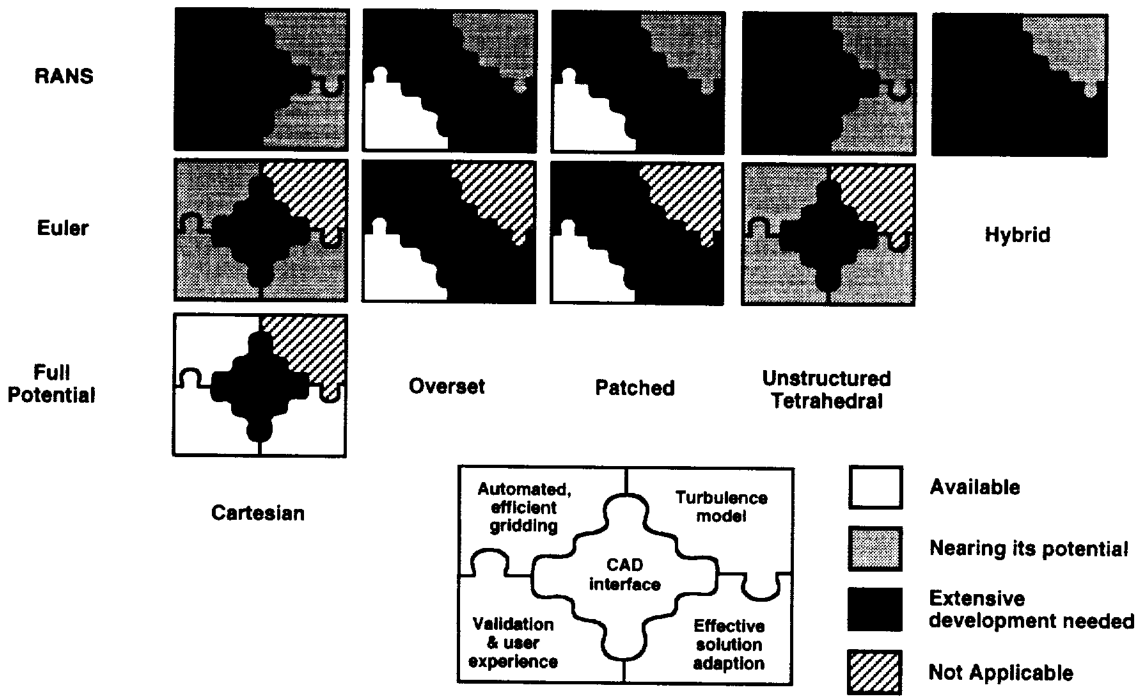
Assessment of Current Technology



CFD software has traditionally been used by very specialized engineers and researchers. Lately, there has been some effort to make codes usable by people less experienced in CFD. Although progress has been made, most applications are still carried out by CFD specialists.

The most usable codes are those which have been available for a long time and have seen the most development or those which have achieved the most automation through extensive development or inherent capability. Experience with the grid generation procedure and the flow solver's limitations is still necessary for effective results. Less well-developed or well-validated techniques require more effort in grid generation, more understanding of the modeling in the flow solver, and/or more acceptance of uncertainty in the results. Several important methods still require considerable research before they can be routinely applied.

Missing Pieces



No CFD method for complex configurations has a simple process path at this time. Despite the advances in standardization by NASA-IGES, the CAD/surface grid interface is still very cumbersome. Overset and patched methods lack automated gridding and are not conducive to efficient solution adaption. Navier-Stokes methods need attention in many areas; hybrid methods may be able to overcome the inherent inefficiency of unstructured Cartesian or tetrahedral grids, yet offer more automation of grid generation than overset or patched methods.

Conclusions

•No one method is the best for all problems. Select the strategy which meets the deadline with the available resources and addresses the important physics of the problem.

•To address a wide range of problems effectively, a full range of tools must be available.

•There is a long process-time gap between Euler and Navier-Stokes analysis.

- •Unstructured methods have done well for Euler problems but have not been satisfactorily demonstrated for viscous 3D problems.

- •Structured methods for a class of problems can be automated for quick turnaround; generalized fast structured capability still does not exist.

- •Hybrid methods could be the answer, if they combine the best of each method.

•Fast Euler methods are attractive from a process-time standpoint, but users must be aware of approximate physics.

•For complicated or detailed geometry, CAD-to-grid is still a major bottleneck.