Numerical CFD Simulation and Test Correlation in a Flight Project Environment

K. K. Gupta[*], S. F. Lung[**] and A. H. Ibrahim[***]

[*]NASA Armstrong Flight Research Center, Edwards, CA
[**]Jacobs Technology Inc., Edwards, CA
[***]Norfolk State University, Norfolk, VA

NASA Armstrong Flight Research Center

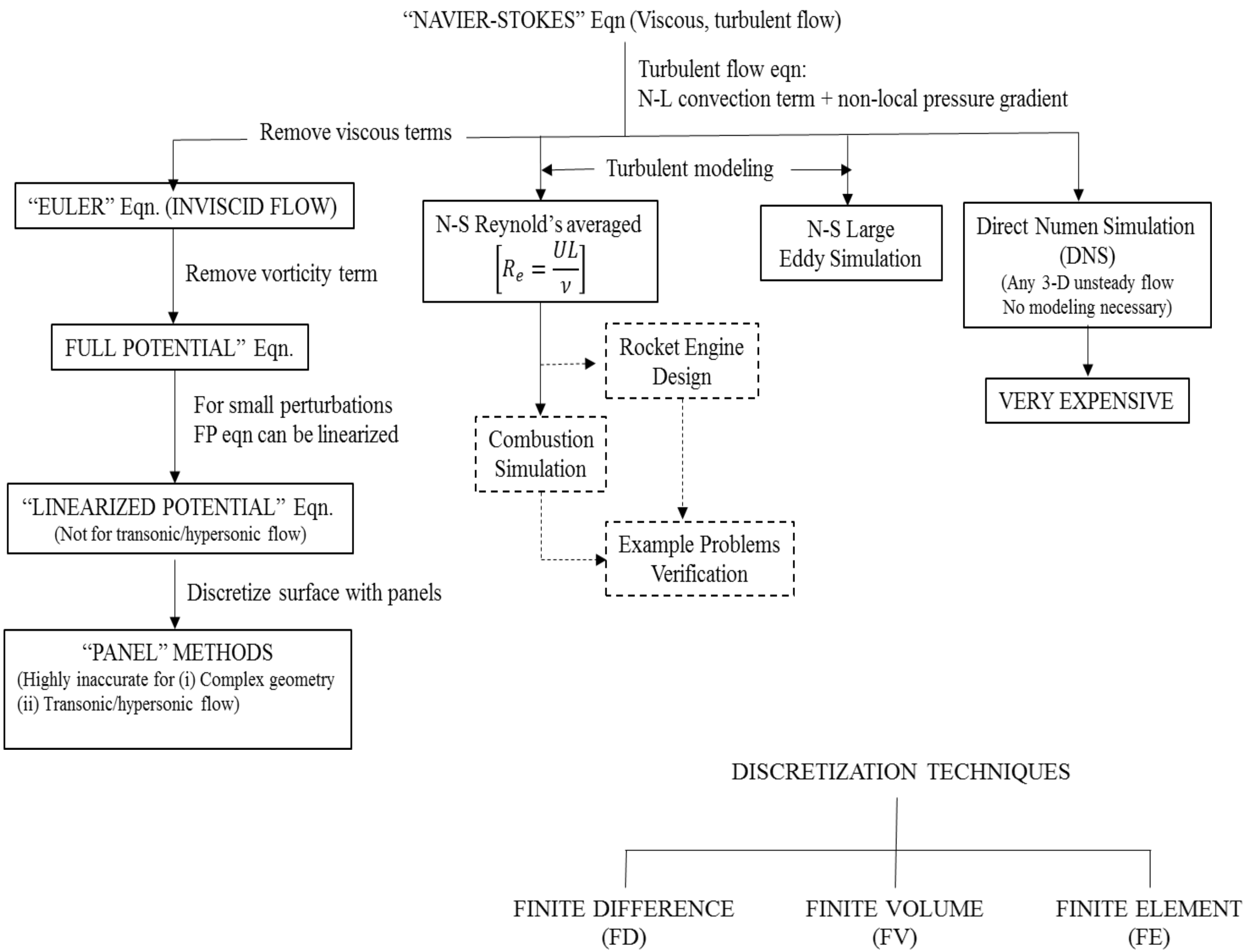
Outline

- INTRODUCTION
- FINITE ELEMENT FORMULATION
- FINITE ELEMENT SOLUTION PROCEDURE
- NUMERICAL EXAMPLE
- RESULTS COMPARISON
- CONCLUSIONS

Introduction

- Develop in-house CFD codes to support NASA Armstrong flight research projects
  - Hyper X/X-43 project
  - Adaptive Compliant Trailing Edge (ACTE) project
- Validate the in-house CFD codes
  - Compare with other commercially available CFD codes
  - Compare with wind tunnel and flight test data

Summary of Flow Equations



Finite Element Formulation

- Navier-Stokes equation:

$$\frac{\partial \boldsymbol{v}}{\partial t} + \frac{\partial \boldsymbol{f_i}}{\partial x_i} + \frac{\partial \boldsymbol{g_i}}{\partial x_i} = f_b \quad i = 1, 2, 3$$

where

$$\boldsymbol{v} = [\rho \quad \rho u_j \quad \rho E]^T, \qquad j = 1,2,3$$
$$f_j = [\rho u_j \quad (\rho u_i u_j + p\delta_{ij}) \quad u_j(p + \rho E)]^T, \qquad j = 1,2,3$$
$$E_j = \left[0 \quad \sigma_{ij} \quad \left(u_i \sigma_{ij} + k\frac{\partial T}{\partial x_j}\right)\right]^T$$
$$f_b = [0 \quad f_{b_i} \quad u_i f_{b_i}]^T$$
$$\sigma_{ij} = \mu\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_l}{\partial x_l}\delta_{ij}\right] \qquad l = 1,2,3$$

- Nondimensionalized for numerical calculation, $\boldsymbol{g_j}$ becomes:

$$g_j = [0 \quad \sigma_{ij} \quad (u_i \sigma_{ij} - q_j)]$$

And

$$\sigma_{ij} = \frac{\mu}{Re}\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_l}{\partial x_l}\delta_{ij}\right]$$
$$q_j = \frac{1}{RePr}\frac{\partial T}{\partial x_j}$$

Finite Element Formulation (Cont.)

- Taylor's expansion of the solution $\boldsymbol{v}(x,t)$ in the time domain:

$$\Delta \boldsymbol{v} = -\Delta t \left[\frac{\partial \boldsymbol{f_i}}{\partial x_i} + \frac{\partial \boldsymbol{g_i}}{\partial x_i} \right]_{(t)}$$

- Applying Galerkin's spatial idealization $\boldsymbol{v} = \boldsymbol{a}\widetilde{\boldsymbol{v}}$, the flow equation can be expressed as

$$\boldsymbol{M}\Delta\widetilde{\boldsymbol{v}} = -\Delta t \left[\frac{\partial u_i}{\partial x_i} \boldsymbol{M} + \boldsymbol{K} \right] \widetilde{\boldsymbol{v}} - \Delta t \left(\hat{\boldsymbol{f}}_1 + \hat{\boldsymbol{f}}_2 \right) + \Delta t \widehat{\boldsymbol{R}} + \Delta t \ [\boldsymbol{K}_\sigma + \boldsymbol{f}_\sigma]$$

- A novel two-step solution procedure is adopted for the flow equation, the inviscid solution being augmented with the viscous term and stabilized with artificial dissipation terms. Assuming

$$\Delta\widetilde{\boldsymbol{v}} = \widetilde{\boldsymbol{v}}_{\boldsymbol{n+1}} - \widetilde{\boldsymbol{v}}_{\boldsymbol{n}}$$

Then

$$\left[\left(1 + \frac{\Delta t}{2} c \right) \boldsymbol{M} + \frac{\Delta t}{2} \boldsymbol{K} \right] \widetilde{\boldsymbol{v}}_{n+1} = \left[\left(1 - \frac{\Delta t}{2} c \right) \boldsymbol{M} - \frac{\Delta t}{2} \boldsymbol{K} \right] \widetilde{\boldsymbol{v}}_n + \Delta t \boldsymbol{R}$$

Or

$$[\boldsymbol{M}_+]\widetilde{\boldsymbol{v}}_{n+1} = [\boldsymbol{M}_-]\widetilde{\boldsymbol{v}}_n + \Delta t \boldsymbol{R}$$

Let

$$\boldsymbol{M}_+ = \boldsymbol{D}_+ + \boldsymbol{M'}_+$$

The viscous stress tensor and heat flux may then be solved as follows:

Finite Element Solution Procedure

- Step 1: Form

$$[\boldsymbol{D}_+]\tilde{\boldsymbol{v}}_{n+1} = [\boldsymbol{M}_-]\tilde{\boldsymbol{v}}_n - [\boldsymbol{M}'_+]\tilde{\boldsymbol{v}}_{n+1} + \Delta t\boldsymbol{R}$$

- Step 2: Solve $\tilde{\boldsymbol{v}}_{n+1}$ iteratively

$$\tilde{\boldsymbol{v}}_{\boldsymbol{n+1}}^{(\boldsymbol{i+1})} = [\boldsymbol{D}_+\,]^{(-\mathbf{1})}\{[\boldsymbol{M}_-\,]\tilde{\boldsymbol{v}}_{\boldsymbol{n}} - [\boldsymbol{M}'_+\,]\tilde{\boldsymbol{v}}_{\boldsymbol{n+1}}^{(\boldsymbol{i})} + \Delta t(\boldsymbol{R} + \widehat{\boldsymbol{R}} + \boldsymbol{K}_{\boldsymbol{\sigma}} + \boldsymbol{f}_{\boldsymbol{\sigma}}\,)\}$$

- Step 3: If $\left\|\tilde{\boldsymbol{v}}_{n+1}^{(\boldsymbol{i+1})}\right\| \neq \text{EPS1}\left\|\tilde{\boldsymbol{v}}_{n+1}^{(\boldsymbol{i})}\right\|$ go to Step 2.

- Step 4: If $\left\|\tilde{\boldsymbol{v}}_{n+1}^{(\boldsymbol{i+1})}\right\| \neq \text{EPS2}\left\|\tilde{\boldsymbol{v}}_{n+1}^{(\boldsymbol{i})}\right\|$ go to Step 1.

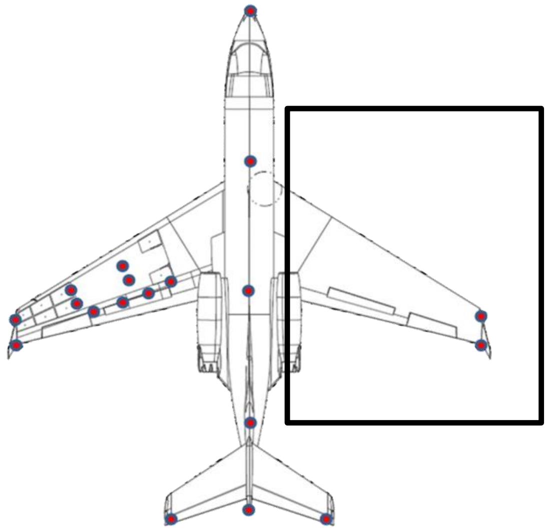
- Step 5: Repeat Steps 1 to 4 NITER times until desired convergence is achieved, that is until $\tilde{\boldsymbol{v}}_{n+1} \approx \tilde{\boldsymbol{v}}_n$; EPS1 and EPS2 are suitable convergence criteria factors, specified by the users.

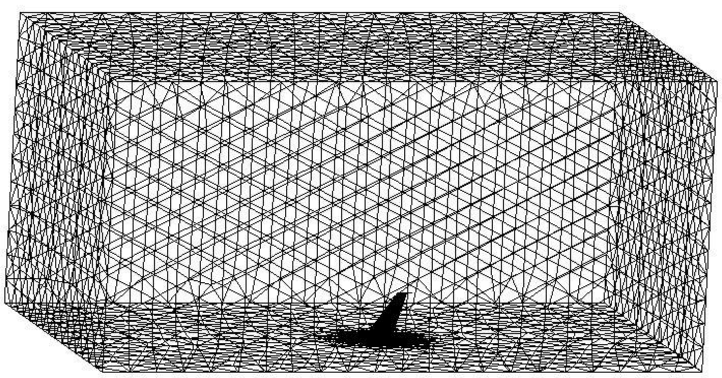
Note: The iterative process in Step 2 requires a small number of steps, usually 1, and achieves a stable, convergent solution. In regions of high pressure gradients, artificial dissipation term is applied to prevent oscillations near discontinuities. This is implemented by incorporating pressure-switched diffusion coefficients as appropriate.

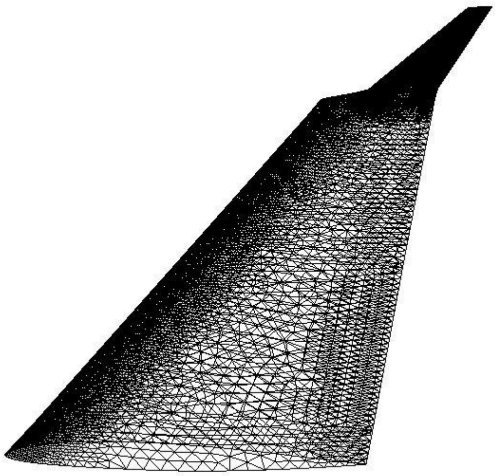
This procedure is adopted in the STARS-CFDSOL code that enables effective solution of the Naviar-Stokes equation in most flight regimes

Numerical Example

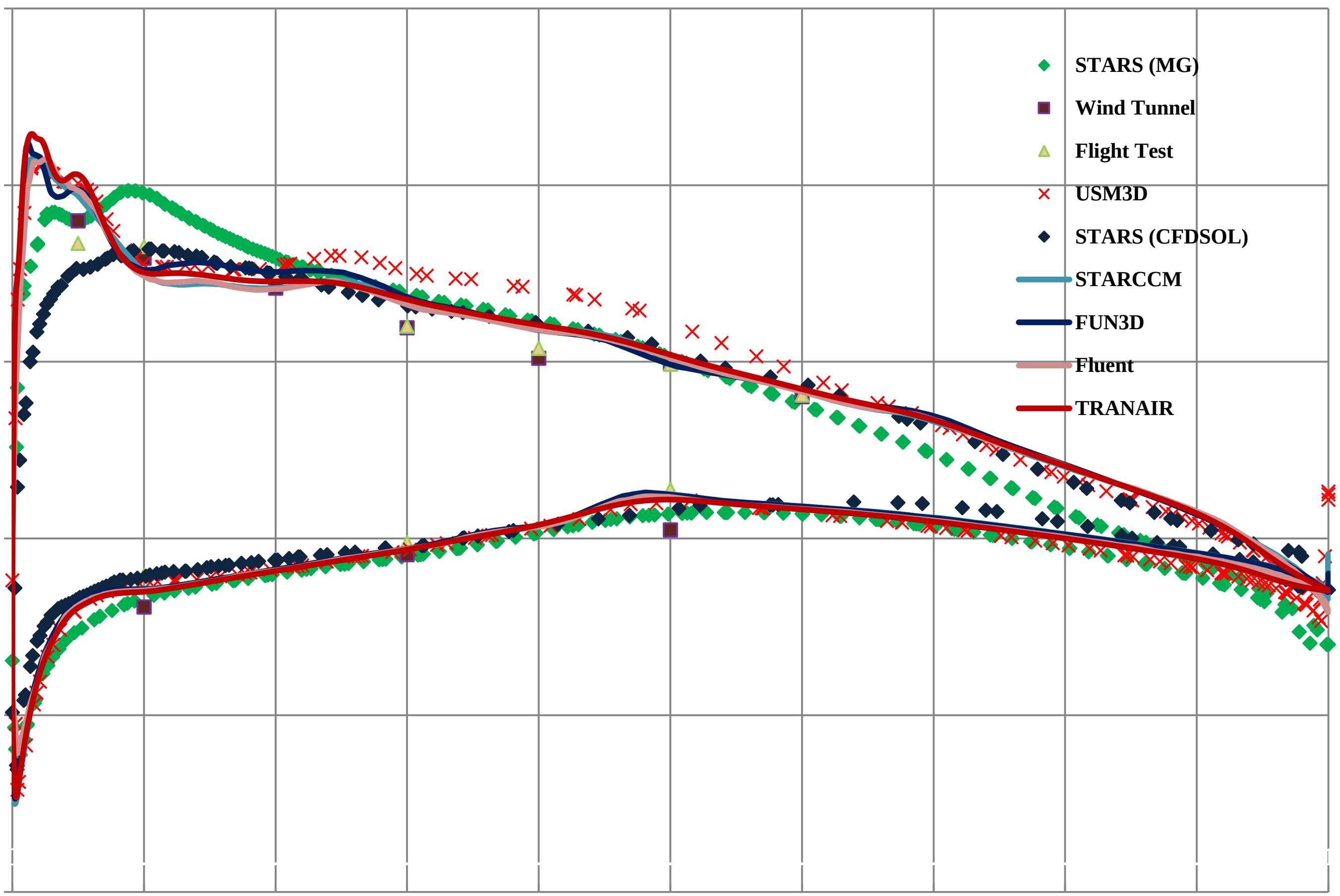
- Gulfstream III (GIII) Wing finite element discretization
  - 15,538 grid 31,072 elements on wing surface
  - 226,444 grid 1,256,879 million elements for the solution domain
- Flight condition:
  - Mach 0.701,
  - Angle of attack 3.92





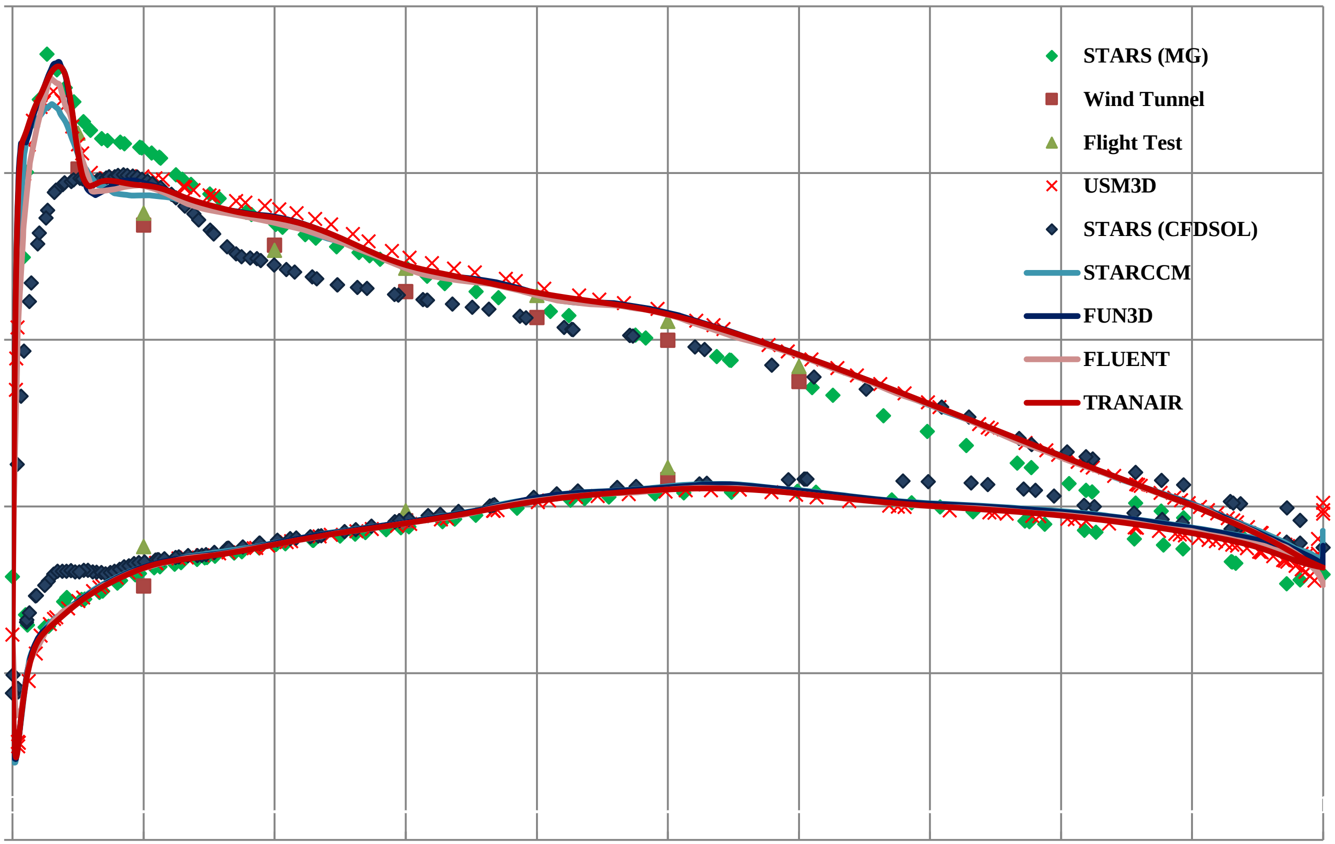


Pressure Distributions



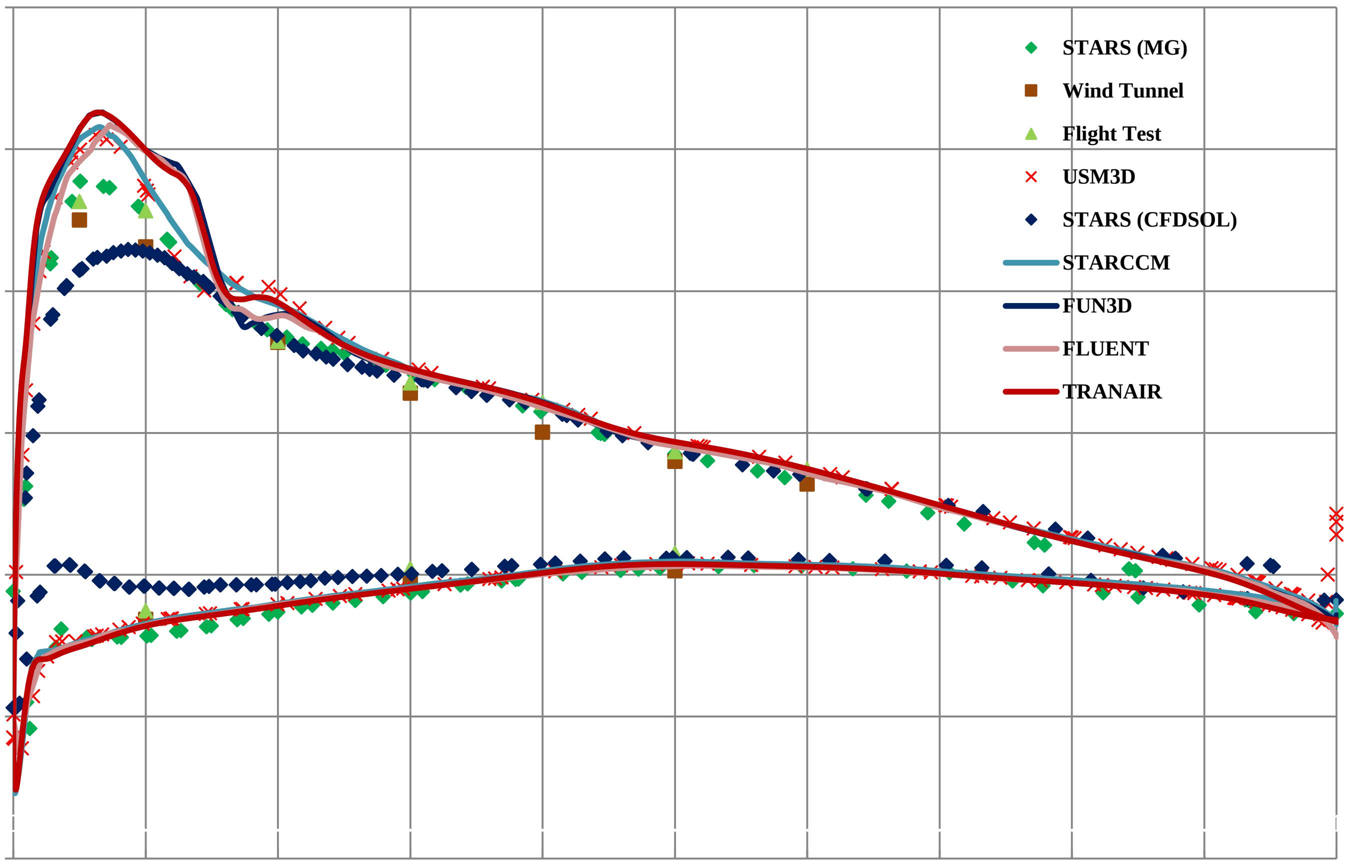
C_p plot at span station 145 (368.3 cm spanwise location)

Pressure Distributions



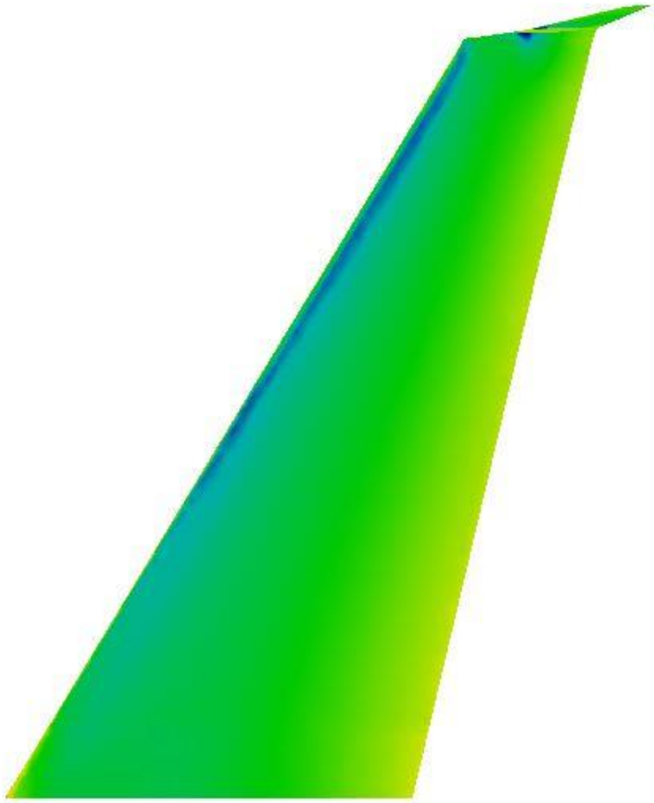
C_p plot at span station 230 (584.2 cm spanwise location)

Pressure Distributions

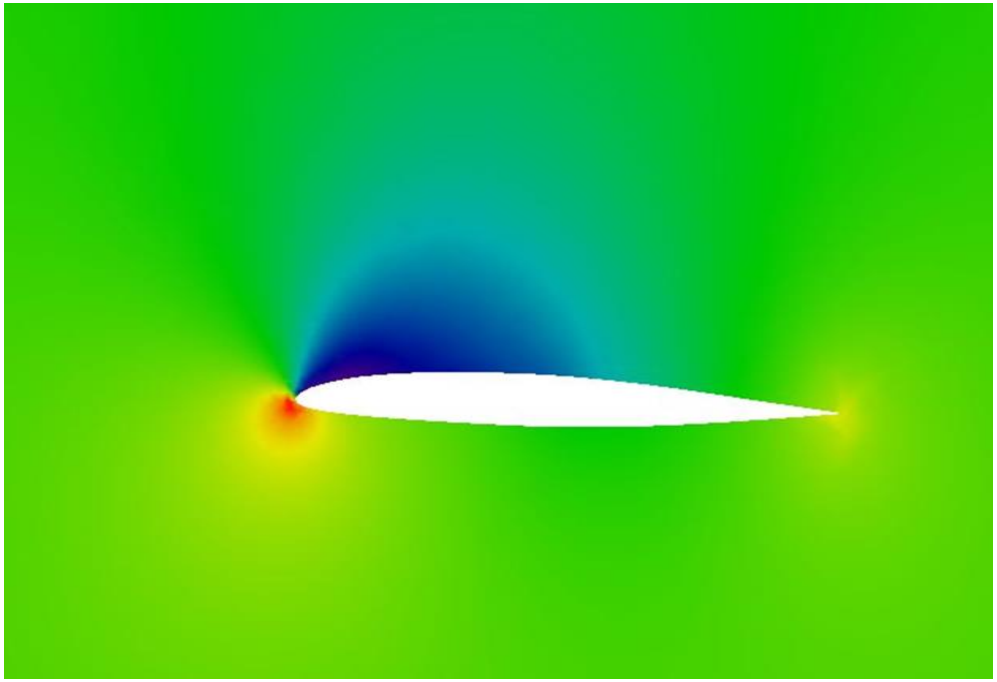


C_p plot at span station 385 (1003.5 spanwise location)

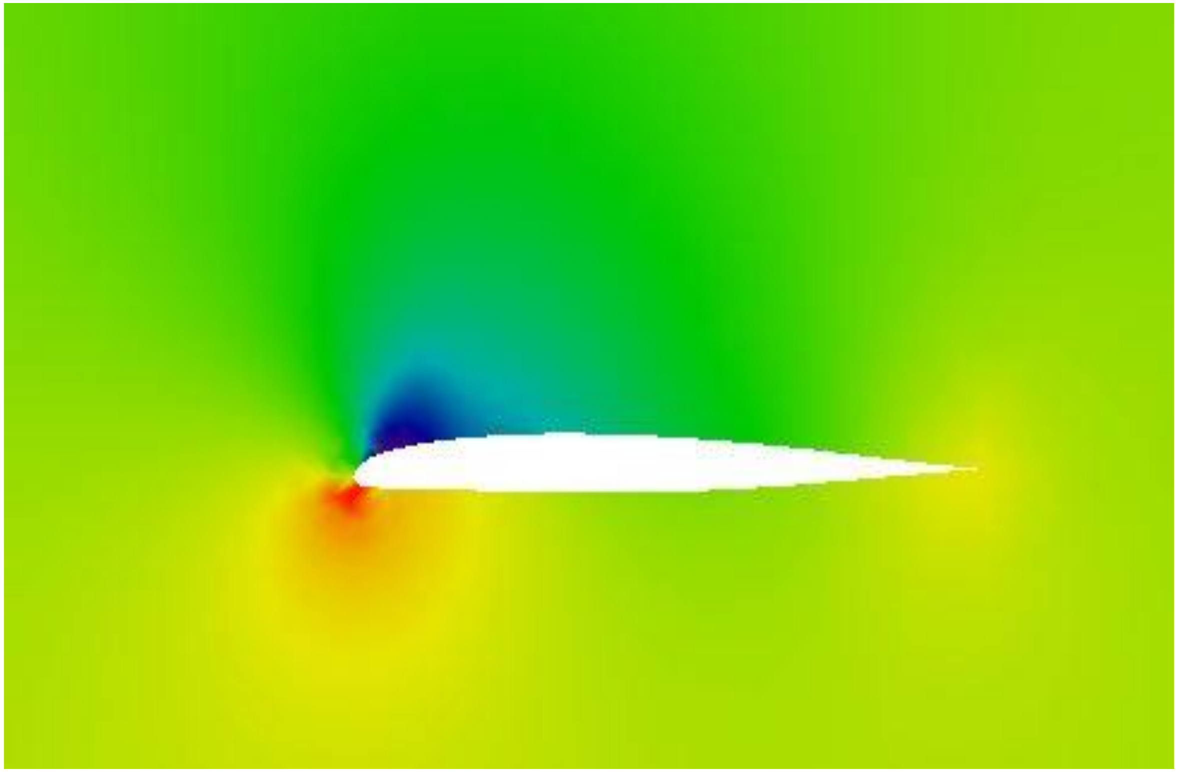
C_p distribution on wing surface



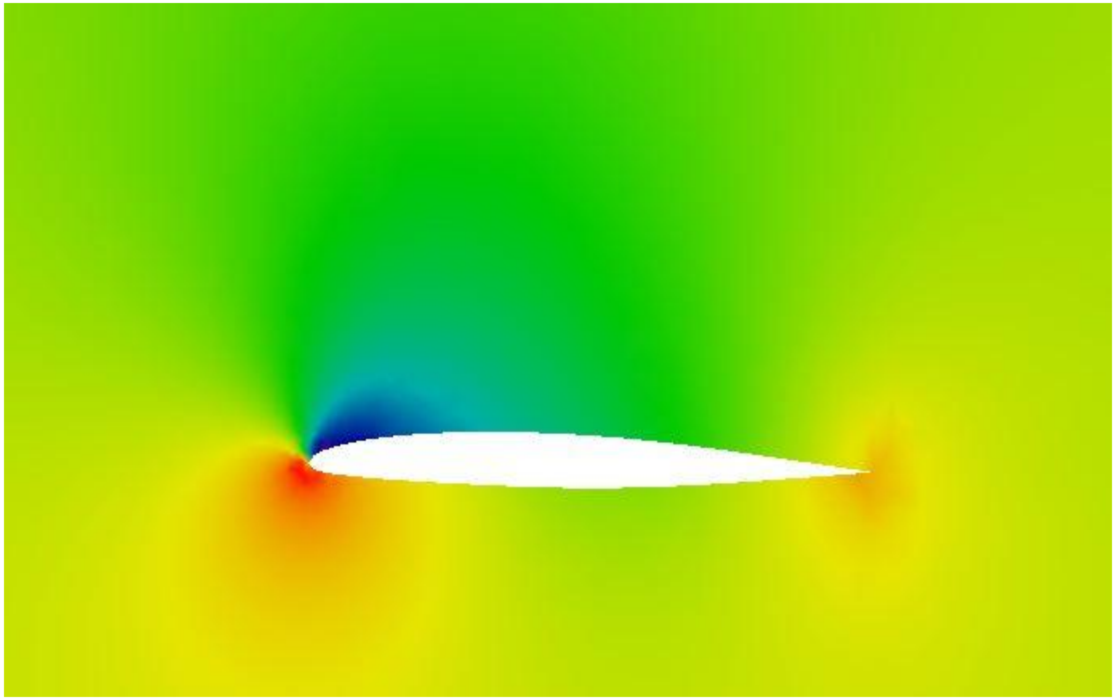
C_p at station 145



C_p at station 230



C_p at station 385



CFD Codes Comparison

CFD Solver	Flow Equation	Platform	No. of CPU	Total CPU time	Grid Size	Note
STARCCM++	RANS, finite volume, K-omega SST turbulence	Cluster	~80	6hrs., 40min (533 CPU hours) - 3000 iterations	7.2M polyhedra/prismatic for half model without T-tail	number of processers is an estimate, and the time is an estimate for that number of processors
Fluent	RANS, Finite Volume, K-Omega SST Turbulence	Cluster	32	16 hrs., (512 CPU hours)	8.12 M tetrahedral/prismatic for half model without T-tail	Time is approximate
STARS (MG)	Euler, finite element	Dell M620 8GB Ram, 64 bit	1 Intel Core i7 @2.67 GHz	2.8 hrs. ,(100 steps, 25 inner cycles)	1.2 M Tetrahedrons for wing only	
STARS (CFDSOL)	Full N-S, finite element	Dell M620 8GB Ram, 64 bit	1 Intel Core i7 @2.67 GHz	13.8 hrs., (10000 steps)	2.8 M Tetrahedrons for wing only	
USM3D	Full N-S, finite volume	Mac 64 bit	2 CPUs	16 hrs.	1.9 M cells for half model without T-tail	
FUN3D	RANS, Finite Volume with turbulence model	Cluster	196	6 hrs.	20.4 M nodes for half model without T-tail	
TRANAIR++	Full potential + viscosity (boundary layer)	Linux Workstation	1 CPU	2hrs., 28min	1.7M cells for full model	

Concluding Remarks

- This paper presents detailed comparison of solutions of the GIII aircraft wing obtained by a number of commercially available CFD codes as well as two NASA AFRC in-house codes that use a finite element discretization employing unstructured grids.

- Finite element formulation of the novel in-house CFD code (CFDSOL) is also presented.

- Each of the codes shows reasonable correlation with flight and wing tunnel test data; CFDSOL and MG codes appears to be rather close to the two test results.

- Use of a single CPU to derive solutions testifies to their cost effectiveness.

CFDSOL Flow Chart