Computational Fluid Dynamics
Past, Present and Future

Prof. Antony Jameson

Department of Aeronautics & Astronautics
– Aerospace Computing Laboratory –
Stanford University
jameson@baboon.stanford.edu
Outline

I. The History of CFD
   - Van Leer’s View
   - Emergence of CFD
   - Multi-disciplinary Nature of CFD
   - Hierarchy of Governing Equations
   - 50 Years of CFD
   - Advances in Computer Power

II. Complexity of CFD

III. Author’s Experience

IV. Usage of CFD – Boeing’s Experience

V. Usage of CFD – Airbus’ Experience

VI. Current & Future Trends

VII. Overview of Numerical Methods

VIII. The FR Methodology

IX. Applications

X. Structural LES Modeling

XI. LES Computations

XII. Summary and Conclusions
The History of CFD

History of CFD in Van Leer’s View

Top level: Jay Boris, Vladimir Kolgan, Bram van Leer, Antony Jameson
Ground level: Richard Courant, Kurt Friedrichs, Hans Lewy, Robert MacCormack, Philip Roe, John von Neumann, Stanley Osher, Amiram Harten, Peter Lax, Sergei Godunov
The History of CFD

Emergence of CFD

- In 1960 the underlying principles of fluid dynamics and the formulation of the governing equations (potential flow, Euler, RANS) were well established
- The new element was the emergence of powerful enough computers to make numerical solution possible – to carry this out required new algorithms
- The emergence of CFD in the 1965–2005 period depended on a combination of advances in computer power and algorithms.

Some significant developments in the ‘60s:
- birth of commercial jet transport – B707 & DC-8
- intense interest in transonic drag rise phenomena
- lack of analytical treatment of transonic aerodynamics
- birth of supercomputers – CDC6600
Multi-Disciplinary Nature of CFD

- Mathematics
- Computer Science
- Aeronautical Engineering
- Fluid Mechanics
- Numerical Wind Tunnel

The History of CFD
The History of CFD

Hierarchy of Governing Equations

I. Linear Potential (1960s)
  - Inviscid, Irrotational
  - Linear

II. Nonlinear Potential (1970s)
  - + Nonlinear

III. Euler (1980s)
  - + Rotation

IV. RANS (1990s)
  - + Viscous

Increasing Complexity
More Accurate Flow Physics
Decreasing Computational Costs
50 Years of CFD

- **1960–1970: Early Developments**
  Riemann-based schemes for gas dynamics (Godunov), 2nd-order dissipative schemes for hyperbolic equations (Lax-Wendroff), efficient explicit methods for Navier-Stokes (MacCormack), panel method (Hess-Smith)

  type-dependent differencing (Murman-Cole), complex characteristics (Garabedian), rotated difference (Jameson), multigrids (Brandt), complete airplane solution (Glowinsky)

- **1980–1990: Euler and Navier-Stokes Equations**
  oscillation control via limiters (Boris-Book), high-order Godunov scheme (van Leer), flux splitting (Steger-Warming), shock capturing via controlled diffusion (Jameson-Schmit-Turkel), approximate Riemann solver (Roe), total variation diminishing (Harten), multigrids (Jameson, Ni), solution of complete airplane (Jameson-Baker-Weatherill)

- **1990–2000: Aerodynamic Shape Optimization**
  adjoint based control theory

- **2000–2010: Discontinuous Finite Element Methods**
  Discontinuous Galerkin, Spectral Difference, Flux Reconstruction, etc.
# The History of CFD

## Advances in Computer Power

<table>
<thead>
<tr>
<th>Year</th>
<th>Machine Type</th>
<th>Performance</th>
<th>Clock Speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>1970</td>
<td>CDC6600</td>
<td>1 Megaflops</td>
<td></td>
</tr>
<tr>
<td>1980</td>
<td>Cray 1</td>
<td>100 Megaflops</td>
<td></td>
</tr>
<tr>
<td>1994</td>
<td>IBM SP2</td>
<td>10 Gigaflops</td>
<td></td>
</tr>
<tr>
<td>2007</td>
<td>Linux Clusters</td>
<td>100 Teraflops</td>
<td></td>
</tr>
<tr>
<td>2007</td>
<td>(affordable) Box Cluster in my house</td>
<td>2.5 Gigaflops</td>
<td>2.5 × 10⁹</td>
</tr>
<tr>
<td>2009</td>
<td>HP Pavilion Quadcore Notebook</td>
<td>1 Gigaflops</td>
<td>10⁹</td>
</tr>
<tr>
<td>2011</td>
<td>MacBook Pro Quadcore Laptop</td>
<td>2.5 Gigaflops</td>
<td>2.5 × 10⁹</td>
</tr>
</tbody>
</table>
Outline

I. The History of CFD

II. Complexity of CFD
   - The Cost of the DoF
   - Grid Size for an Aircraft Wing
   - Complexity of CFD in the ‘70s
   - Complexity of CFD in the ‘80s
   - CFD Complexity for Turbulence

III. Author’s Experience

IV. Usage of CFD – Boeing’s Experience

V. Usage of CFD – Airbus’ Experience

VI. Current & Future Trends

VII. Overview of Numerical Methods

VIII. The FR Methodology

IX. Applications

X. Structural LES Modeling

XI. LES Computations

XII. Summary and Conclusions
Fluid dynamic problems involve polynomials with large $N$ and fairly large $p$.

Complexity of Fluid Dynamic Simulations - Explicit Schemes

- With $N \approx n^3$ mesh points in 3D and explicit time stepping, each time step requires $O(n^3)$ operations.
- The time step of a stable scheme is proportional to the mesh interval $h$ divided by the wave speed, and $h \approx 1/n$, giving complexity $Cn^4 \approx N^{4/3}$ with a constant $C$ depending on the algorithm.

Complexity of Fluid Dynamic Simulations - Implicit Schemes

- An implicit scheme requires matrix inversion at each time step with complexity $NB^2$ where $B$ is the bandwidth $\approx n^2$, so the cost of a step is $O(n^7)$.
- The time step is not limited by the mesh interval, so the number of time steps is independent of $n$, giving total complexity $\approx n^7$. 
Grid Size for a Transport Aircraft Wing

- 32 cells in the boundary layer
- 512 cells around the wing to limit the mesh aspect ratio (to about 1000)
- 256 cells spanwise

Total: $512 \times 64 \times 256 = 8,388,608$ cells
The complexity of a 3D prediction of transonic flow is $O(n^4)$ and reasonable accuracy can be obtained with $n \approx 100$

Calculations could be completed in $O(10^8)$ operations with a CDC 6600 which could achieve $\approx 10^6$ flops

Thus a useful 3D calculation might be possible in $O(10^2)$ seconds

The author recognized this in 1971

Actually FLO22 (Jameson and Caughey), which was the first program which could actually predict transonic flow over a swept wing with engineering accuracy, required about 10,000 seconds for a solution
Complexity of CFD in the ‘80s

- 800,000 mesh cells for a viscous mesh around a wing
- 5,000 flops per solution step using FLO107
- 300 steps for the solution to converge
- \( (8 \times 10^5) \times (5 \times 10^3) \times (3 \times 10^2) = 1.2 \times 10^{12} \)

Roughly \(10^{12}\) flops for RANS simulation on 0.8 million mesh cells

With a 1 Gigaflop computer, solution takes about 1,000 seconds...

... About 400 seconds with a 2011 MacBook Pro quadcore at 2.5 Gflops
For a turbulent flow with a Reynolds number $Re$, the length scale of the smallest eddies relative to the integral length scale $\approx Re^{-3/4}$ (Kolmogorov, 1943)

With a comparable time step, the complexity of the simulations $\approx Re^3$

For a jumbo jet such as the Airbus A380, $Re \approx 10^8$

Direct Numerical Simulation (DNS) of the flow over the A380 has a complexity $\approx 10^{24}$ operations

With a Petaflop computer (IBM Roadrunner, 2008), DNS of the A380 has a complexity of about $10^9$ seconds

About 30 Years!
Outline

I. The History of CFD
II. Complexity of CFD
III. Author’s Experience
   - CFD Code Development
   - FLO and SYN Codes
   - Wing Optimization Using SYN107
IV. Usage of CFD – Boeing’s Experience
V. Usage of CFD – Airbus’ Experience
VI. Current & Future Trends
VII. Overview of Numerical Methods
VIII. The FR Methodology
IX. Applications
X. Structural LES Modeling
XI. LES Computations
XII. Summary and Conclusions
CFD Code Development

  solution of inverse problem by conformal mapping (SYN1), solution of 2D potential flow by conformal mapping (FLO1), 2D transonic potential flow using rotated difference scheme (FLO6), first transonic potential flow solution for a swept wing (FLO22), 3D potential flow in general grid with trilinear isoparametric elements (FLO27), multigrid solution of 2D transonic potential flow (FLO36)

- **1980–1990: Euler & Navier-Stokes Equations**
  solution of 3D Euler (FLO57), multigrid solution of 3D Euler (FLO67), multigrid solution of 2D Euler (FLO82), first solution of Euler equations for a complete aircraft with tetrahedral meshes (FLOPLANE), cell-vertex and cell-centered schemes for 3D Navier-Stokes (FLO107)

- **1990–2000: Aerodynamic Shape Optimization**
  airfoil design via control theory using 2D Euler (SYN83), wing design using 3D Euler (SYN88), airfoil design using 2D Navier-Stokes (SYN103), wing design using 3D Navier-Stokes (SYN107), aerodynamic design of complete aircraft with tetrahedral mesh (SYNPLANE), viscous flow solution on arbitrary polyhedral meshes (FLO3XX)

  high-order discontinuous finite element methods for unsteady compressible Navier-Stokes equations on unstructured meshes (Spectral Difference Method, Energy Stable Flux Reconstruction Method)
**Author’s Experience**

**FLO and SYN Codes**

<table>
<thead>
<tr>
<th>YEARS</th>
<th>CODES</th>
<th>AIRPLANES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1970</td>
<td>FLO1, 2, SYN1</td>
<td>Canadair: Challenger (FLO 22)</td>
</tr>
<tr>
<td>1971-1973</td>
<td>FLO 6</td>
<td>Regional Jet (FLO 22)</td>
</tr>
<tr>
<td>1975</td>
<td>FLO 22</td>
<td>Global Express (airplane)</td>
</tr>
<tr>
<td>1977</td>
<td>FLO 27</td>
<td>Northrop: B2 (FLO 22)</td>
</tr>
<tr>
<td>1979</td>
<td>FLO 36</td>
<td>F23 (FLO 57)</td>
</tr>
<tr>
<td>1981</td>
<td>FLO 52, 57</td>
<td>Boeing: 737-500 (FLO 27-28 incorporated in Boeing A488 software)</td>
</tr>
<tr>
<td>1984</td>
<td>FLO 62, 67</td>
<td>747-400 (FLO 27-28 incorporated in Boeing A488 software)</td>
</tr>
<tr>
<td>1985</td>
<td>AIRPLANE</td>
<td>757 (FLO 27-28 incorporated in Boeing A488 software)</td>
</tr>
<tr>
<td>1985</td>
<td>FLO 82, 87</td>
<td>767 (FLO 27-28 incorporated in Boeing A488 software)</td>
</tr>
<tr>
<td>1988</td>
<td>FLO 97, 107</td>
<td>777 (FLO 27-28 incorporated in Boeing A488 software)</td>
</tr>
<tr>
<td>1989</td>
<td>SYN 36</td>
<td>787 (FLO 27-28 incorporated in Boeing A488 software)</td>
</tr>
<tr>
<td>1991</td>
<td>UFLO 82, 87</td>
<td>McDonnell-Douglas: C17 (FLO 22), MD11 (FLO 22, airplane)</td>
</tr>
<tr>
<td>1993-1995</td>
<td>SYN 87, 88</td>
<td>MD12 (FLO 67, airplane), MDXX (SYN 88)</td>
</tr>
<tr>
<td>1997</td>
<td>SYN 107</td>
<td>MDHSCT (FLO 67, airplane), MD90 (FLO 27 incorporated in dactran10)</td>
</tr>
<tr>
<td>2001</td>
<td>FLO 82-SGS, FLO 88-SGS</td>
<td>MD95: later Boeing 717 (FLO 22, FLO 67)</td>
</tr>
<tr>
<td>2003</td>
<td>SYNPLANE</td>
<td>Beech: Premier (SYN 87 MB), Horizon (SYN 87 MB)</td>
</tr>
<tr>
<td>2003-2006</td>
<td>FLO-3xx, SYN-3xx</td>
<td>Embraer: 190 (SYN 88)</td>
</tr>
</tbody>
</table>
State of the Art Wing Design Process in 2 Stages, starting from Garabedian-Korn Airfoil and NASA Common Research Model
Outline

I. The History of CFD

II. Complexity of CFD

III. Author’s Experience

IV. Usage of CFD – Boeing’s Experience
   • Impact of CFD on Configuration Lines & Wind Tunnel Testing
   • Impact of CFD on B737-300 Program
   • Computational Methods at Boeing
   • CFD Contributions to B787

V. Usage of CFD – Airbus’ Experience
   • CFD Development for Aircraft Design
   • Block-Structured RANS Capability: FLOWer
   • Unstructured RANS Capability: TAU
   • Numerical Flow Simulation
   • CFD Contributions to A380

VI. Current & Future Trends

VII. Overview of Numerical Methods

VIII. The FR Methodology

IX. Applications

X. Structural LES Modeling

XI. LES Computations

XII. Summary and Conclusions
Usage of CFD – Boeing's Experience

Impact of CFD on Configuration Lines & Wind Tunnel Testing

- NASA Tech
  - PANAIR
  - FLO22
  - A411

- Boeing Tools
  - A502
  - A488

- Boeing Products
  - 767
  - 757
  - 737-300

- 1980 state of the art
  - 77

- Modern close coupled nacelle installation, 0.02 Mach faster than 737-200

- 1985

- Cartessian Grid Tech.
  - TRANAIR

- 1990

- TLNS3D

- 1995

- HSR & IWD
  - TLNS3D-MB
  - OVERFLOW

- 2000

- CFL3D
  - OVERFLOW

- CFL3D/ZEUS
  - OVERFLOW

- CFD++

- Unstructured Adaptive Grid
  - 3-D N-S

- 2005

- 787

- Number of Wings Tested
  - 1980: 38
  - 1995: 18
  - 2005: 11

50% Reduction in Wind Tunnel Testing!

- 1980

- 1995

- 2005

- Boeing

- A. Jameson

AMS Seminar, October 30th, 2012
Impact of CFD on B737-300 Program


20 Years of wind tunnel based development indicated nacelles cannot be placed too close to the wing without excessive drag

Joint CFD/Wind Tunnel Studies unlock the secret of nacelle/wing interference drag

707/CFM56 Design & Flight Test validated CFD concepts

737-300 Program initially rejected due to high cost of increasing landing gear length

Go Ahead Roll Out Certification

Initial Studies 737-300 Program

McDonnell-Douglas MD-80 Go Ahead

Without the understanding gained from CFD there would not have been a 737-300 Program!

Walt Gillette
Manager, 737 Aerodynamics - then
Vice President, 787 Engineering – retired

5000+ Additional Sales!
Computational Methods at Boeing

**TRANAIR:**
- Full Potential with directly coupled Boundary Layer
- Cartesian solution adaptive grid
- Drela lag-dissipation turbulence model
- Multi-point design/optimization

**Navier-Stokes Codes:**
- CFL3D – Structured Multiblock Grid
- TLNS3D – Structured Multiblock Grid, Thin Layer
- OVERFLOW – Overset Grid

**N-S Turbulence Models:**
- S-A Spalart-Allmaras
- Menter’s $k$-$\omega$ SST
MEGAFLOW / MEGADESIGN
- National CFD Initiative (since 1995)

Development & validation of a national CFD software for complete aircraft applications which

- allows computational aerodynamic analysis for 3D complex configurations at cruise, high-lift & off-design conditions
- builds the basis for shape optimization and multidisciplinary simulation
- establishes numerical flow simulation as a routinely used tool at DLR and in German aircraft industry
- serves as a development platform for universities
Efficient simulation tool for configurations of moderate complexity

- advanced turbulence and transition models (RSM, DES)
- state-of-the-art algorithms
  - baseline: JST scheme, multigrid
  - robust integration of RSM (DDADI)
- chimera technique for moving bodies
- fluid / structure coupling
- design option (inverse design, adjoint)

FLOWer-Code

- Fortran
- portable code
- parallelization based on MPI
Unstructured RANS Capability: TAU

Tool for complex configurations
- hybrid meshes, cell vertex / cell centered
- high-level turbulence & transition models (RSM, DES, linear stability methods)
- state-of-the-art algorithms (JST, multigrid, ...)
- local mesh adaptation
- chimera technique
- fluid / structure coupling
- continuous/discrete adjoint
- extensions to hypersonic flows

TAU-Code
- unstructured database
- C-code, Python
- portable code, optimized for cache hardware
- high performance on parallel computer
Usage of CFD – Airbus' Experience

Numerical Flow Simulation

Relation CFD / wind tunnel

- CFD
- wind tunnel

improvements
algorithms & hardware

unstructured
hybrid grids

number of simulations
> 30,000

CFD cost effective alternative
Usage of CFD – Airbus’ Experience

CFD Contribution to A380

- Frequent use
- Moderate use
- Growing use

- High Speed Wing Design
- Ice Prediction
- Flutter Prediction
- Flow Control Devices (VG/Strakes)
- Low Speed Wing Design
- Cabin Ventilation
- Cabin Noise
- Fuselage Design
- Cockpit/Avionics Ventilation
- Performance Prediction
- Powerplant Integration
- Nacelle Design
- Inlet Design
- Wing Tip Design
- External Noise Sources
- Handling Quality Data
- Ground Effect
- Static Deformation
- Pack Bay Thermal Analysis
- Aero Loads Data
- Engine Core Compartment
- Thrust Reverser Design
Outline

I. The History of CFD
II. Complexity of CFD
III. Author’s Experience
IV. Usage of CFD – Boeing’s Experience
V. Usage of CFD – Airbus’ Experience
VI. Current & Future Trends
   - The Current Status of CFD
   - The Future of CFD (?)
   - Large-Eddy Simulation

VII. Overview of Numerical Methods
VIII. The FR Methodology
IX. Applications
X. Structural LES Modeling
XI. LES Computations
XII. Summary and Conclusions
The Current Status of CFD

- Worldwide commercial and government codes are based on algorithms developed in the ‘80s and ‘90s
- These codes can handle complex geometry but are generally limited to 2nd order accuracy
- They cannot handle turbulence without modeling
- Unsteady simulations are very expensive, and questions over accuracy remain
CFD has been on a plateau for the past 15 years

- Representations of current state of the art:
  - Formula 1 cars
  - Complete aircrafts

- The majority of current CFD methods are not adequate for vortex dominated and transitional flows:
  - Rotorcraft
  - High-lift systems
  - Formation flying
Large-Eddy Simulation

The number of DoF for an LES of turbulent flow over an airfoil scales as $Re_c^{1.8}$ (resp. $Re_c^{0.4}$) if the inner layer is resolved (resp. modeled).

Rapid advances in computer hardware should make LES feasible within the foreseeable future for industrial problems at high Reynolds numbers. To realize this goal requires

- high-order algorithms for unstructured meshes (complex geometries)
- Sub-Grid Scale models applicable to wall bounded flows
- massively parallel implementation

Chapman (1979), AIAA J. 17(12)
Outline

I. The History of CFD
II. Complexity of CFD
III. Author’s Experience
IV. Usage of CFD – Boeing’s Experience
V. Usage of CFD – Airbus’ Experience
VI. Current & Future Trends

VII. Overview of Numerical Methods
   • Typical Requirements of CFD
   • Classic Numerical Methods
   • A Review of the Literature

VIII. The FR Methodology
IX. Applications
X. Structural LES Modeling
XI. LES Computations
XII. Summary and Conclusions
Traditional numerical schemes for engineering problems are too dissipative and do not provide sufficient accuracy for LES and DNS.

- **Accuracy:** solution must be right
- **Small numerical dissipation:** unsteady flow features
- **Unstructured grids:** complex geometries
- **Numerical flux:** wave propagation problems
- **High resolution capabilities:** transitional and turbulent flows
- **Efficiency:** code parallelism
- ...
Overview of Numerical Methods

**Classic Numerical Methods**

**Finite Difference**
- Structured
- High-order
- Numerical flux

**Finite Volume**
- Unstructured
- Low-order
- Numerical flux

**Continuous FE**
- Unstructured
- High-order
- No numerical flux

**Discontinuous FE**
- Unstructured
- High-order
- Numerical flux
A Review of the Literature

Past Research on DG Schemes:

Recent Research:
Attempts to reduce complexity and avoid quadrature:
- Spectral Difference (SD) scheme by Kopriva & Kolas [1996], Liu, Vinokur & Wang [2006]
- Nodal Discontinuous Galerkin (NDG) scheme by Atkins & Shu [1998], Hesthaven & Warburton [2007]
- Flux Reconstruction (FR) scheme by Huynh [2007,2009]
Outline

I. The History of CFD
II. Complexity of CFD
III. Author’s Experience
IV. Usage of CFD – Boeing’s Experience
V. Usage of CFD – Airbus’ Experience
VI. Current & Future Trends
VII. Overview of Numerical Methods
VIII. The FR Methodology
   - The Flux Reconstruction Scheme
   - The FR Scheme Graphically Illustrated
   - Energy Stability of the FR Scheme
   - A Family of Energy Stable Schemes
   - Accuracy & Numerical Dissipation
   - High-Order Boundaries
   - Computational Cost
IX. Applications
X. Structural LES Modeling
XI. LES Computations
XII. Summary and Conclusions
The FR Methodology

Introduction

- The following presentation emphasizes development of Huynh's FR approach, and energy stability
- Energy stability analysis versus Fourier stability analysis
  - Energy method is more general and rigorous
  - Energy method enables stability proofs for all orders of accuracy
  - Energy method applies to non-uniform meshes
  - Fourier analysis provides more detailed information about the distribution of dispersive and diffusive errors
The FR Methodology

The Flux Reconstruction Scheme

The solution is locally represented by Lagrange polynomial of degree $n - 1$ on the solution points:

$$u_h = \sum_{j=1}^{n} u_j l_j(x)$$

$$f_h^D = \sum_{j=1}^{n} f_j^D l_j(x)$$

The flux is discontinuous and needs to be corrected in a suitable way

$$\Delta_L = \tilde{f}_L - f_h^D (-1)$$

$$\Delta_R = \tilde{f}_R - f_h^D (1)$$

$$g_L(-1) = 1, \quad g_L(1) = 0$$

$$g_R(1) = 1, \quad g_R(-1) = 0$$

The continuous flux is obtained from the discontinuous counterpart by adding the correction functions of degree $n$ weighted by the flux corrections

$$f_h^C = \sum_{j=1}^{n} f_j^D l_j(x) + g_L(x)\Delta_L + g_R(x)\Delta_R$$

The continuous flux is finally differentiated at the solution points and the solution is advanced in time

$$\frac{\partial u_i}{\partial t} + \left[ \sum_{j=1}^{n} f_j^D \frac{dl_j}{dx}(x_i) + \Delta_L \frac{dg_L}{dx}(x_i) + \Delta_R \frac{dg_R}{dx}(x_i) \right] = 0$$

The FR Methodology

The FR Scheme Graphically Illustrated

The solution is locally represented by Lagrange polynomial of degree \( n-1 \) on the \( n \) solution points:

\[
    u_h = \sum_{j=1}^{n} u_j l_j(x)
\]

The discontinuous flux is constructed

\[ f_h^D = \sum_{j=1}^{n} f_j^D l_j(x) \]

The FR Methodology

The FR Scheme Graphically Illustrated

Solution is evaluated at element boundaries

\[ u_L = \sum_{j=1}^{n} u_j l_j (-1) \]

\[ u_R = \sum_{j=1}^{n} u_j l_j (+1) \]

The FR Methodology

The FR Scheme Graphically Illustrated

The **common** interface flux is computed from multiply defined values at each interface (FV-type numerical flux such as approximate Riemann flux)

\[
\tilde{f}_L, \quad \tilde{f}_R
\]

The common interface flux is computed from multiply defined values at each interface (FV-type numerical flux such as approximate Riemann flux)
The FR Methodology

The FR Scheme Graphically Illustrated

Correction functions of degree $n$ are introduced

$$g_L(-1) = 1, \quad g_L(1) = 0$$

The correction functions are scaled

$$\Delta_L = \tilde{f}_L - \tilde{f}_D (-1)$$

The FR Methodology

The FR Scheme Graphically Illustrated

The correction is added to the discontinuous flux

\[ f_h^* = \sum_{j=1}^{n} f_j^D l_j(x) + g_L(x) \Delta L \]
The right boundary is corrected the same way

\[ g_R(1) = 1, \quad g_R(-1) = 0 \]
The FR Methodology

The FR Scheme Graphically Illustrated

The correction is scaled...

$$\Delta_R = \tilde{f}_R - f_h^D (+1)$$
The FR Methodology

The FR Scheme Graphically Illustrated

And added to the discontinuous flux

\[ f_h^C = \sum_{j=1}^{n} f_j^D l_j(x) + g_L(x)\Delta_L + g_R(x)\Delta_R \]
The FR Methodology

The FR Scheme Graphically Illustrated

Total approximate continuous flux

\[ f^C_h = \sum_{j=1}^{n} f^D_j l_j(x) + g_L(x) \Delta_L + g_R(x) \Delta_R \]
The divergence of the flux is evaluated at the solution points

\[
\frac{\partial f^C}{\partial x}(x_i) = \sum_{j=1}^{n} f_j^D \frac{dl_j}{dx}(x_i) + \Delta_L \frac{dg_L}{dx}(x_i) + \Delta_R \frac{dg_R}{dx}(x_i)
\]
The solution is advanced in time

\[
\frac{\partial u_i}{\partial t} + \left[ \sum_{j=1}^{n} f_j^D \frac{dl_j}{dx}(x_i) + \Delta_L \frac{dg_L}{dx}(x_i) + \Delta_R \frac{dg_R}{dx}(x_i) \right] = 0
\]

The FR Methodology

Energy Stability of the FR Scheme

The FR method defines a family of energy stable schemes in the norm

$$||U^{\delta D}||_{p,2} = \left[ \sum_{n=1}^{N} \int_{x_n}^{x_{n+1}} (U^{\delta D}_n)^2 + \frac{c}{2} (J_n)^{2p} \left( \frac{\partial p U^{\delta D}_n}{\partial x^p} \right)^2 \, dx \right]^{1/2}$$

The schemes have the form

$$\frac{\partial u_i}{\partial t} + \left[ \sum_{j=1}^{n} f_j^D \frac{dl_j}{dx}(x_i) + \Delta_L \frac{dh_L}{dx}(x_i) + \Delta_R \frac{dh_R}{dx}(x_i) \right] = 0$$

where the correction functions in terms of Legendre polynomials are

$$h_L = \frac{(-1)^p}{2} \left[ L_p - \left( \frac{\eta_p(c)L_{p-1} + L_{p+1}}{1 + \eta_p(c)} \right) \right]$$

$$h_R = \frac{(+1)^p}{2} \left[ L_p + \left( \frac{\eta_p(c)L_{p-1} + L_{p+1}}{1 + \eta_p(c)} \right) \right]$$

with a single parameter $c$

$$\eta_p(c) = \frac{c(2p + 1)(a_p p!)^2}{2}$$

The FR Methodology

A Family of Energy Stable Schemes

Nodal DG:

\[ c = 0 \quad \Rightarrow \quad \eta_p = 0 \]

\[ g_L = \frac{(-1)^p}{2} \left[ L_p - L_{p+1} \right], \quad g_R = \frac{(+1)^p}{2} \left[ L_p + L_{p+1} \right] \]

Spectral Difference:

\[ c = \frac{2p}{(2p+1)(p+1)(a_p)!^2} \quad \Rightarrow \quad \eta_p = \frac{p}{p+1} \]

\[ g_L = \frac{(-1)^p}{2} (1-x)L_p, \quad g_R = \frac{(+1)^p}{2} (1+x)L_p \]

G2 Scheme by Huynh [2007]:

\[ c = \frac{2(p+1)}{(2p+1)p(a_p)!^2} \quad \Rightarrow \quad \eta_p = \frac{p+1}{p} \]

\[ g_L = \frac{(-1)^p}{2} \left[ L_p - \frac{(p+1)L_{p-1} + pL_{p+1}}{2p+1} \right], \quad g_R = \frac{(+1)^p}{2} \left[ L_p + \frac{(p+1)L_{p-1} + pL_{p+1}}{2p+1} \right] \]
The FR Methodology

Accuracy & Numerical Dissipation

Temporal Mixing-Layer

N=5: 100 × 200 × 10 DoF

N=6

N=2

60 × 60 DoF

N=5: 100 × 200 × 10 DoF
The FR Methodology

Accuracy & Numerical Dissipation

N=6, 60×60×12 DoF

Vorticity magnitude

Iso-Q
High-Order Boundaries

Cylinder: \( N=4, \ 32 \times 32, \) linear vs. cubic

Sphere: \( N=3, \) linear vs. quadratic

The FR Methodology

Computational Cost

10000 iterations on my Mac

- Scheme: SD
- Time integration: RK3
- Nº of elements: $7 \times 7 \times 1$
- Nº of processors: 1

Parallelism:
interprocessor interfaces never involve more than one level of exchanged data
compact stencil

- The cost per DoF scales linearly with $N$ (beyond $N = 5$)
- The optimum is obtained for $N = 4$ (viscous) and $N = 5$ (inviscid)
Outline

I. The History of CFD
II. Complexity of CFD
III. Author’s Experience
IV. Usage of CFD – Boeing’s Experience
V. Usage of CFD – Airbus’ Experience
VI. Current & Future Trends
VII. Overview of Numerical Methods
VIII. The FR Methodology
IX. Applications
   - Transitional Flow over SD7003 Airfoil
   - Study of Flapping Wing Sections
   - Flapping Wing Aerodynamics
   - Flow Over Spheres
X. Structural LES Modeling
XI. LES Computations
XII. Summary and Conclusions
### Applications

#### Transitional Flow over SD7003 Airfoil

<table>
<thead>
<tr>
<th></th>
<th>Freestream</th>
<th>Separation $x_{sep}/c$</th>
<th>Transition $x_{tr}/c$</th>
<th>Reattach. $x_r/c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radespiel et al.</td>
<td>0.08%</td>
<td>0.30</td>
<td>0.53</td>
<td>0.64</td>
</tr>
<tr>
<td>Ol et al.</td>
<td>0.10%</td>
<td>0.18</td>
<td>0.47</td>
<td>0.58</td>
</tr>
<tr>
<td>Galbraith Visbal</td>
<td>0%</td>
<td>0.23</td>
<td>0.55</td>
<td>0.65</td>
</tr>
<tr>
<td>Uranga et al.</td>
<td>0%</td>
<td>0.23</td>
<td>0.51</td>
<td>0.60</td>
</tr>
<tr>
<td>Present ILES*</td>
<td>0%</td>
<td>0.23</td>
<td>0.53</td>
<td>0.64</td>
</tr>
</tbody>
</table>

Experiments in green

---

*1.7×10^6 DoF

---

![Iso-Q colored by Ma](image)

**SD scheme, N=4**

**Re=6×10^4, AoA=4°, 2.2×10^7 DoF**

---

Study of Flapping Wing Sections

SD, 2D, N=5 on deforming grid

Experiment (Jones, et al.)

NACA0012, Re=1850, Ma=0.2, St=1.5, $\omega=2.46$, $h=0.12c$

Flapping Wing Aerodynamics

Iso-Entropy colored by Ma

Flapping NACA0012, Re=2000, SD N=5, $4.7 \times 10^6$ DoF

Wing-Body, Re=5000, SD N=4, $2.1 \times 10^7$ DoF

Flow Over Spheres

Flow over a spinning sphere,
Re=300, Ma=0.2

Flow over a sphere,
Re=10000, Ma=0.2

Outline

I. The History of CFD
II. Complexity of CFD
III. Author’s Experience
IV. Usage of CFD – Boeing’s Experience
V. Usage of CFD – Airbus’ Experience
VI. Current & Future Trends

VII. Overview of Numerical Methods
VIII. The FR Methodology
IX. Applications

X. Structural LES Modeling
   ● Discrete Filtering Operators
   ● Explicit Filtering in the SD Element

XI. LES Computations
   ● Wall-Resolved Turbulent Channel
   ● Wall-Modeling Strategy
   ● Wall-Modeled Turbulent Channel
   ● Flow past a Square Cylinder

XII. Summary and Conclusions
The filtering operator for the 1D standard element is defined as

$$\overline{\phi}_s = \sum_{i=1}^{N} w_i^s \phi_i, \quad (s = 1, \ldots, N)$$

The kernel of the above discrete filter can be written as

$$\hat{G}_s(k) = \sum_{i=1}^{N} w_i^s \exp(-j\beta_i^s k\Delta), \quad \text{with} \quad \beta_i^s = \frac{\xi_i - \xi_s}{\Delta}$$

$\Delta=1/N$ is assumed to be the actual resolution within the SD element.
Explicit Filtering in the SD Element

Key issues:
- non-uniform and staggered distribution of points
- the filter stencil shall not lie across elements
- filter width shall be prescribed and constant

LES Computations

Wall-Resolved Turbulent Channel Flow

$Re_\tau = 180$

$Re_\tau = 395$

$Re_\tau = 590$

LES Computations

Wall-Resolved Turbulent Channel Flow

\( Re_\tau = 180 \) 
\( \Delta^+ : 38, 2-10, 19 \)

\( Re_\tau = 395 \) 
\( \Delta^+ : 39, 1-40, 26 \)

\( Re_\tau = 590 \) 
\( \Delta^+ : 60, 3-33, 26 \)

**LES Computations**

Wall-Modeling Strategy

Breuer and Rodi (1996)
LES Computations

Wall-Modeled Turbulent Channel Flow

**Reτ = 590**  \((\Delta^+: 58, 24-47, 58)\)

**Reτ = 2000**  \((\Delta^+: 98, 22-102, 98)\)

Wall-Modeled Turbulent Channel Flow

$Re_{\tau} = 590$

$Re_{\tau} = 2000$

$u$ at $y^+ = 100$

Wall-Resolved

Wall-Modeled
LES Computations

Flow Past Square Cylinder: Re = 21400

- Time integration: RK3
- Nº of elements: 35760 (2.3 \times 10^6 \text{ DoF})
- Grid dimensions: $21D \times 12D \times 3.2D$
- Reynolds: 21400
- Mach: 0.3
- Statistics: 16 $T_0$
Flow past a Square Cylinder: \( \text{Re}_D = 21400 \)

iso-\( Q \) colored by velocity

Preliminary results (work in progress)

\[
\begin{align*}
\langle u \rangle / U_b & \quad \langle u' \rangle / U_b^2 \\
\langle u' \rangle / U_b^2 & \quad \langle u' \rangle / U_b^2
\end{align*}
\]
The early development of CFD in the Aerospace Industry was primarily driven by the need to calculate steady transonic flows: *this problem is quite well solved*

CFD has been on a plateau for the last 15 years with 2nd-order accurate FV methods for the RANS equations almost universally used in both commercial and government codes which can treat complex configurations.

These methods cannot reliably predict complex separated, unsteady and vortex dominated flows.

Ongoing advances in both numerical algorithms and computer hardware and software should enable an advance to LES for industrial applications within the foreseeable future.

Research should focus on high-order methods with minimal numerical dissipation for unstructured meshes to enable the treatment of complex configurations.

Eventually DNS may become feasible for high Reynolds number flows, *hopefully with a smaller power requirement than a wind tunnel*.
Acknowledgement

The current research is a combined effort by

- **Postdocs**: Charlie Liang, Peter Vincent, Guido Lodato
- **Ph.D. students**: Sachin Premasuthan, Kui Ou, Patrice Castonguay, David Williams, Yves Allenau, Lala Li, Manuel Lopez, and Andy Chan

It is made possible by the support of

- the **Airforce Office of Scientific Research** under grant FA9550-10-1-0418 by Dr. Fariba Fahroo
- the **National Science Foundation** under grants 0708071 and 0915006 monitored by Dr. Leland Jameson
A Review of the Literature from ACL


5. Lodato, G., P. Castonguay, and A. Jameson, Structural LES modeling with high-order spectral difference schemes. In *Annual Research Briefs* (Center for Turbulence Research, Stanford University, 2011)


Questions & Answers

Thank you for listening