FLIGHTSTREAM UNSTRUCTURED SURFACE VORTICITY SOLVER

NASA VSP WORKSHOP 2014
San Luis Obispo, California
Research in Flight: Company Background

- Research in Flight is an Alabama LLC formed in 2012 for the purposes of commercializing work in the general area of volumetrically meshless solutions.

- The primary venture of the Research in Flight Company is currently the development of the software known as FlightStream™

- Key Personnel and Contact Information:
  - Roy J. Hartfield, Jr., PhD
    roy.hartfield@researchinflight.com
    334 444 8523
  - Vivek Ahuja, PhD
    vivek.ahuja@researchinflight.com
    334 332 6078
  - Website: www.researchinflight.com
  - Physical Address:
    1919 North Ashe Court
    Auburn, AL, 36830
Introduction

- Need for optimization based design processes is only growing
- Optimization runs typically require thousands of automated design evaluations
- For aerospace applications, this is a problem of minimizing time versus maintaining fidelity of solution and validity of physics
- Determination of aerodynamic loads is a major requirement for many such applications
- Typical evaluation choices include empirical methods, panel methods, CFD and experiments
# Aerodynamic analysis tools

<table>
<thead>
<tr>
<th></th>
<th>Empirical formulations</th>
<th>Potential flow methods</th>
<th>Computational Fluid Dynamics</th>
<th>Experimental methods</th>
</tr>
</thead>
<tbody>
<tr>
<td>Complexity of input data/geometry</td>
<td>Low</td>
<td>Intermediate</td>
<td>Heavy</td>
<td>Heavy</td>
</tr>
<tr>
<td>Run time for single design</td>
<td>Low</td>
<td>Intermediate</td>
<td>Intermediate/Heavy</td>
<td>Heavy</td>
</tr>
<tr>
<td>Solution fidelity</td>
<td>Intermediate</td>
<td>Intermediate</td>
<td>Good</td>
<td>Good</td>
</tr>
<tr>
<td>Automation possibilities for optimization</td>
<td>Very high</td>
<td>high</td>
<td>Low</td>
<td>Very low</td>
</tr>
<tr>
<td>Range of physics</td>
<td>Limited</td>
<td>Limited</td>
<td>High</td>
<td>High</td>
</tr>
<tr>
<td>Cost of process</td>
<td>Very low</td>
<td>Low</td>
<td>High</td>
<td>Very high</td>
</tr>
</tbody>
</table>
Potential flow solvers

- Inside the subsonic flow regime, the advantages of potential flow methods make them highly popular and effective to design studies:
  - **Surface meshes only**
    - Saves substantial run time
    - Reduces potential areas that could adversely affect solution fidelity
  - **Stability**
    - Potential flow methods are also mostly stable
    - Few exceptions can exist:
      - In wake relaxation
      - Thin bodies
  - **Quick solution convergence**
    - Even on large, finely refined surface meshes
Potential flow solvers

- Biggest drawback of potential-flow methods, is the surface mesh:
  - Pressure-field solvers:
    - Able to use an unstructured surface mesh to evaluate flows
    - But solver is based on evaluating the pressure field
      - Emulation Navier-Stokes solver philosophy
      - Using commercial CFD is more economical!
  - Vorticity-field solvers:
    - Restricted to structured meshes with known U-V mapped surfaces
    - Applicability of the Kutta-Joukowski lift equation requires:
      - Knowledge of “bound” and “trailing” vortex segments
      - Needs them aligned at the correct angles to the flow
    - Retain advantages of speed and fidelity
Motivation

- “........paradigm shifts in meshing technology (i.e., cut cell methods, strand grids, meshless methods) may lead to revolutionary advances in simulation capabilities.”


Panel codes were not the precise target of this comment; nevertheless, we believe that Flightstream represents a step in the direction of reducing the amount of meshing required to get accurate load predictions.
Motivation

- *What is needed is a Vorticity-based potential-flow solver which:*

  - Retains advantages of philosophical approach of Laplacian solvers:
    - Fidelity in subsonic regime
    - Speed of solver
    - Robust against geometry defects

  - Able to work with unstructured surface meshes:
    - Opens wide spectrum surface geometries from all CAD/CAE platforms to analysis
    - Creates higher quality meshes
    - Creates lower face count meshes
    - Greater flexibility to an optimizer during MDO
Unstructured mesh solver: why?

- The good news:
  - Structured meshes are the primary meshing choice for Vorticity-solvers:
    - The “bound” and “trailing” vortices are aligned in the U-V mapping
    - Meshes are easier to generate and easier to control
Unstructured mesh solver: why?

The bad news:

“Bad quality” meshes for regions of high curvature
- U-V mapping unable to “recognize” and “react”
- Facets are long, needle-shaped and poor quality
- U-V mapping forces edge-lengths to collapse to zero in physical space
- These poor facets create solver oscillations and divergence

Larger mesh size:
- Enforced U-V mapping creates very small facets in regions not needing them
- These facets add up substantially to the overall mesh cell-count while failing to add any usable quality to the geometry
- Much larger cell counts equal much higher solver run times:

\[ \text{Computational time} \propto \text{facet}^2 \]

Aerospace and 3D modeling Industry uses unstructured meshes:
- Higher spectrum geometries/models can be tested
- Newer applications
- Direct connectivity with standard CAD/CAE packages
Unstructured mesh solver: why?

Typical structured mesh over a high curvature surface

Region of lesser curvature forced to have higher facets because of UV mapping

Long needle-shaped triangles to satisfy UV mapping
Much higher cell count
Unstructured mesh solver: why?

- Unique advantages for potential-flow solvers:
  - Meshes can be locally modified:
    - Physics-defined refinements
    - Solver-defined de-featuring
  - High gradient topology easily captured
  - No enforced U-V mappings:
    - Poor quality cells can be deleted, improved or refined
    - Mesh size is reduced
    - Faster solution convergence and reduced instabilities
    - Control over topological gradients
  - Local application of boundary and external physics
Unstructured mesh solver: why?

<table>
<thead>
<tr>
<th></th>
<th>Structured</th>
<th>Unstructured</th>
<th>Reduction (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of facets</td>
<td>1404</td>
<td>648</td>
<td>53.85</td>
</tr>
<tr>
<td>Solver run time (Seconds)</td>
<td>48.88</td>
<td>5.71</td>
<td>88.32</td>
</tr>
<tr>
<td>CL (expected = 0.0)</td>
<td>0.000025</td>
<td>0.000009</td>
<td>64.00</td>
</tr>
<tr>
<td>Convergence (expected = 1E-3)</td>
<td>9E-3</td>
<td>7E-4</td>
<td>92.22</td>
</tr>
<tr>
<td>Solver iterations</td>
<td>20</td>
<td>7</td>
<td>65.00</td>
</tr>
</tbody>
</table>
NASA VSP & FlightStream
Vehicle Sketch Pad: Advantages

- A versatile tool for the aerospace industry

- Rapidly models aircraft configurations without need for traditional CAD packages

- Can be used to design relatively detailed and advanced geometries with full parametric control

- FlightStream and VSP are completely synchronized for rapid solutions:
  - Mesh files from VSP can be directly imported into FlightStream
  - Meshes are usually solver-ready requiring a minimal of repair
  - FlightStream can bypass stitched region faces on the mesh with ease
    - Surface vorticity in FlightStream is generally insensitive to facet shape
  - FlightStream uses output from VSP CompGeom; No surface meshing needed!
  - FlightStream Quad mesher recovers structured face pairs wherever possible
Vehicle Sketch Pad: Quad meshing?

- **FlightStream Quad mesher:**
  - The surface quad mesher recovers pairs of surface face triangles and recovers them to quads
  - Useful for cases where the geometry has origins in structured environments.
    - VSP qualifies
    - Typical CAD software do not qualify. That’s okay too!
  - Leaves areas of unsuitable quad recovery as triangles, such as:
    - Stitched areas of the mesh
  - FlightStream works with a hybrid valence surface mesh environment. Can work with:
    - Triangular faces
    - Quad faces
    - Poly faces
    - All or any of the above
  - User can choose to use quad mesher on one or all surfaces obtained from VSP
Vehicle Sketch Pad: Quad meshing?

Boeing-767 preliminary model from VSP as seen after import into FlightStream

Region of high quad recovery

Same model seen in the FlightStream solver after initialization
Vehicle Sketch Pad: Quad meshing?

- **Quad meshing advantages:**
  - Lower solver-mesh face count; same geometry results in lesser solver run time for convergence

  - Aligned surface faces. Important when encountering very thin surfaces
    - Such as near wing trailing edges
    - Having the upper and lower surfaces aligned leads to greater solver stability

  - Lower mesh-induced surface vorticity “bumps” caused by unstructured triangulation

  - Generally greater solver stability and faster convergence
Vehicle Sketch Pad: Quad meshing?

Quad meshing advantages:

- C-130H
- F-16C
- Boeing Ecojet
- DLR-F6
- Advanced Tiltrotor
- NASA Canard Fighter
- F-35
- Quad mesh

Graph 1: Computation time (seconds) vs. Mesh size (faces)

Graph 2: Time savings (%) vs. Recovery Factor

Graph Key:
- Theoretical
- Structured mesh case
- NASA Canard fighter
- Boeing Ecojet
- DLR-F6
- F-16C
- C-130H
- Advanced Tiltrotor
General Geometry Operations In FlightStream
General Geometry Operations: Import

- FlightStream can import geometries and geometrical CAD data from numerous sources:
  - STL format files
  - TRI format files
  - User created custom U-V mapped surfaces
  - FlightStream Manual Mesher
  - FlightStream Extruder

- VSP geometries are brought in via STL format; users can append various STL files to create assemblies in FlightStream

- New surfaces can be created from imported ones; imported surfaces can be:
  - Preserved
  - Combined
  - United
  - Mirrored
  - Copied
  - Translated
  - Rotated
  - Deleted
  - Exported
General Geometry Operations: U-V maps

- FlightStream allows users to import U-V mapped surfaces generated from any programming software platform such as Matlab, Fortran, C++ etc.

- The U-V mapped meshes are easy to create and often serve as useful analysis and CAD control options.

- 2-D meshes can also be imported and then converted to 3-D within FlightStream.
Mesh Repair In FlightStream
Mesh repair: Diagnostics

- FlightStream works with discrete meshes rather than native CAD data.
- As such, topology and patches can be controlled and analyzed.
- Typical diagnostics of interest for the solver include the feature edges, patch/surface perimeters, free-edges and proximal/pierced face pairs.
Mesh repair: Atomic mesh repair

- When all else fails, FlightStream allows the user to manually repair the mesh wherever needed.

- Operations can be done on:
  - Faces
  - Edges
  - Vertices
  - Patches
Mesh repair: Extruder

- FlightStream Extruder allows users to extrude surfaces between two curves to create a structured mesh.
- Surfaces can also be generated by way of projection to a user-defined plane, or by revolving around an axis.
- Mesh settings for this tool include the sizing, distribution and growth rates.
- The resulting surface is independently added to the assembly and can then be merged with any other existing surface.
- Users can also import curve and edge data from excel spreadsheets or data files.
Mesh repair: Extruder

- Extrusion between curve/edge sets:
  - Users can connect a pair of edge sets either on the mesh or imported in from spreadsheets or data sets
  - Extrusion takes place between a source and target curve set
  - The resulting mesh is unstructured but maintains alignment between the source and target curves
  - A very useful tool that works in conjunction with VSP and any other imported geometry
  - Can be used to re-tessellate any specific section of the overall geometry without affecting other surfaces
Fluid Physics In FlightStream
Once the user has created their surface sets out of their original geometry, the application of physics and boundary conditions can be done.

Steady-flow surface physics in FlightStream can be categorized in the following criteria:

- Facet physics
- Edge physics
- Vertex physics
- Volume physics

Facet, edge and vertex physics can be directly applied on the surfaces and require no additional geometry-level changes.

Volume physics require the user to create volume shapes around the geometry.
Facet Physics: Slip walls

- Facet physics refers to the boundary conditions that are applied to the faces of any given surface.
- Facets could be marked individually or the whole surface could be marked depending on the requirements.
- By default, all surfaces of the mesh are automatically marked as slip-walls unless marked otherwise.
- This means that for most cases, the user need only bring the mesh in and not have to worry at all about facet-based physics.

**Slip walls:**

- **The Neumann condition has been used:**
  - Stipulates that fluid flow normal to the facet must be zero.
  - Applied on each facet’s control point.
  - Allows usage with both thick and thin surfaces.

- **Slip wall conditions cannot carry any boundary layer information:**
  - This decouples the skin-friction analysis.
  - Skin-friction must be evaluated separately.
Facet Physics: Velocity Inlets

- The user can also mark surfaces as velocity inlets to simulate inflow and outflow conditions.

- For example, the compressor fan face of an external pod engine on a commercial airliner would need to be marked as velocity inlets.

- Similarly, sections on the fuselage can be marked as inflow to simulate suction or fluid expulsion effects.

Velocity Inlet equaling stagnation flow

Velocity Inlet equaling free-stream
Facet Physics: Symmetry planes

- For most fixed-wing aircraft geometries in steady flow, a symmetry plane relevant to the fluid flow can be identified.

- In such cases the solver can be allowed to mirror the inductions from one half of the mesh on the other side.

- **Greatly reduced solver workload:**
  - Only needs to be evaluated once at solver initialization phase
  - Removes half of the wake
  - Reduces solver partitions by half

- This routine can save between 30-70% off the overall run-time depending on size of case
  - Larger the mesh size, larger the savings
Edge Physics

- Edge physics in FlightStream refers to the marking of vorticity shedding edges in steady-flow studies.
- The user has the option of marking edges manually for specific faces and surface perimeters or using the auto-detection tool.
- The auto-detector marks the shedding edges for a geometry based on the user-defined free-stream conditions:
  - Uses topology parameters to determine trailing locations.
  - Can be controlled using expert mesh settings if required.
- This auto-detection tool for the unstructured mesh allows FlightStream to work in an automated optimization environment.
- The vorticity is shed from those facet edges that have trailing edges marked for it.
Vertex Physics

- Vertex physics in FlightStream refers specifically to the control of the vorticity strands being shed from marked trailing edges.

- It is sometimes required to terminate wake strands that are within a viscous layer of the flow.

- Such scenarios are typically encountered at the junction of wings and fuselages.

- It is possible to select a vertex on the mesh and mark it to allow the solver to either stabilize the strand coming from that node or to terminate it.

- Such vertex physics modes allow greater solver stability and enhanced user control on the physics.
Volume Physics

- Volumetric flow controls can also be created in FlightStream to simulate super-velocity conditions such as:
  - Downwash of a propeller upstream of a wing
  - Rotation of a turbine blades in front of a stock and housing for wind-turbine.

- Users can create their local coordinate frames and apply volumetric controls to simulate these type of external flow conditions on surfaces of interest.
Modeling Steady State Flows In FlightStream

Case Studies
## Validation cases

- Seven advanced test cases have been investigated in addition to several simple test models:

<table>
<thead>
<tr>
<th>Test case</th>
<th>Source</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>F-16A Fighting Falcon</td>
<td>AGARD conference proceedings, n 242, October 11-13, 1977</td>
<td>Webb, T.S., Kent, D.R.</td>
</tr>
<tr>
<td>DLR-F6</td>
<td>AIAA CFD Drag Prediction Workshop</td>
<td>-</td>
</tr>
<tr>
<td>Rutan Varieze</td>
<td>NASA TP-2382</td>
<td>-</td>
</tr>
<tr>
<td>Advanced General Aviation Wing</td>
<td>NASA TN D-8236</td>
<td>Semi-span / Fowler Flap</td>
</tr>
<tr>
<td>NASA Trap Wing</td>
<td>1&lt;sup&gt;ST&lt;/sup&gt; AIAA CFD High Lift Prediction Workshop (HLPW1)</td>
<td>Full-span / Fowler Flap</td>
</tr>
<tr>
<td>DLR-F11</td>
<td>2&lt;sup&gt;ND&lt;/sup&gt; AIAA CFD High Lift Prediction Workshop (HLPW2)</td>
<td>Full-span / Fowler Flap</td>
</tr>
</tbody>
</table>
Validation Cases: F-16A Fighting Falcon

Case setup:

- Initial geometry contained 4,504 facets with no underlying CAD

- De-featured:
  - Weapons stores and pylons
  - Electronics antennae
  - Gun fairing
  - Closure of engine intakes and exhaust
  - Conversion of horizontal stabilizer to thin surface

- Model Boundary Physics:
  - Trailing-edges for wings and horizontal stabilizers only
  - Intake modeled as Velocity-Inlet
  - Slip walls for surface
  - Symmetry plane for overall geometry
Validation Cases: F-16A Fighting Falcon

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Area (m^2)</td>
<td>27.89</td>
</tr>
<tr>
<td>Mach Number</td>
<td>0.9</td>
</tr>
</tbody>
</table>
Validation Cases: F-18A Hornet

Case setup:

- Initial geometry contained 19,702 facets with no underlying CAD

- De-featured:
  - Weapons stores and pylons
  - Electronics antennae
  - Closure of engine intakes and exhaust
  - Conversion of stabilizers to thin surface

- Model Boundary Physics:
  - Trailing-edges for wings and stabilizers
  - Slip walls for surface and symmetry plane for overall geometry
Validation Cases: F-18A Hornet

Angle-of-Attack 1 Degrees

Vorticity

0.16
0.12
0.08
0.04
0
-0.04
-0.08
-0.12
-0.16
Validation Cases: F-18A Hornet

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Area (m²)</td>
<td>0.035</td>
</tr>
<tr>
<td>Mach Number</td>
<td>0.33</td>
</tr>
</tbody>
</table>
Case setup:

- Geometry obtained from the OpenVSP Hanger
- Model was split along symmetry plane and additional span sections were added in VSP for the main wing
- Overall mesh face count: 2,291 triangles (1,271 quads with 89.04% quad mesher recovery factor)
- Model Boundary Physics:
  - Trailing-edges for wings and stabilizers
  - Slip walls for surface and symmetry plane for overall geometry
Validation Cases: DLR-F6

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Area (m^2)</td>
<td>0.1454</td>
</tr>
<tr>
<td>Mach Number</td>
<td>0.75</td>
</tr>
</tbody>
</table>

![Graph 1](image1.png)

![Graph 2](image2.png)
Validation Cases: Rutan Varieze

Case setup:

- Geometry created in VSP
- Model was split along symmetry plane and additional span sections were added in VSP for the main wing
- Overall mesh face count: 3,109 triangles (1,810 quads with 83.56% quad mesher recovery factor)

- Model Boundary Physics:
  - Trailing-edges for wings and stabilizers
  - Velocity-Inlets for engine intake
  - Slip walls for surface and symmetry plane for overall geometry
Validation Cases: Rutan Varieze

Parameter | Value
--- | ---
Reference Area (ft^2) | 53.6
Mach Number | 0.1

Lift Coefficient vs. Angle of attack (Deg)

- NASA Langley WT, Canard only
- NASA Langley WT, Main Wing only
- NASA Langley WT, Combined
- Decoupled-combined
Advanced General Aviation Wing

- **Geometry:**
  - Symmetrical wing wind-tunnel model
  - Semi-span flap configuration
  - Spoilers attached above flap sections
  - Fowler flap data in both stowed and deployed conditions (0°, 10°, 20°, 30°, 40° flap angles plus associated chordwise translation)
  - Incompressible flow conditions
  - GA(W)-1 airfoil
Advanced General Aviation Wing

Meshing in FlightStream:
- Semi-span model created assuming symmetry plane flow conditions
- Mesh generated using FlightStream extruder to capitalize on zero-sweep spanwise direction and recover maximum surface quads (upwards of 90% recovery)
- Data collected for stowed spoiler configuration only, hence not modeled
- Flap brackets not modeled
- 2,470 surface faces
- 1,238 solver quads
Advanced General Aviation Wing

- Solver run-time conditions:
  - Parallel run conditions on 2 processors
  - ~45 seconds run-time for single AOA condition till solver convergence
  - Relaxed wake conditions on flaps and main wing
  - Symmetry plane boundary applied
Advanced General Aviation Wing

Results:

- Data collected for 10°, 20°, and 30° flap deflection angles
- Angles of attack sweep matched to that in NASA TN D-8236
NASA Trap Wing (AIAA HLPW1 Geometry)

- **Geometry:**
  - Geometry obtained from AIAA 1\textsuperscript{ST} CFD High-Lift Prediction Workshop
  - Full-span flap and slat configuration
  - Slat at 30°, Flap at 25° (plus associated chordwise translation)
  - Near-incompressible (Mach 0.2) flow conditions
Meshing in FlightStream:
- Mesh generated using FlightStream extruder and surface quad mesher (78.25% quad recovery)
- 5,030 surface faces
- 3,062 solver quads
NASA Trap Wing (AIAA HLPW1 Geometry)

- Solver run-time conditions:
  - Parallel run conditions on 2 processors
  - ~150 seconds run-time for single AOA condition till solver convergence
  - Relaxed wake conditions on flaps and main wing
NASA Trap Wing (AIAA HLPW1 Geometry)

- Results:
  - Angles of attack sweep matched to that in AIAA HLPW1 Experimental data
DLR-F11

- **Geometry:**
  - Geometry obtained from AIAA 2\textsuperscript{ND} CFD High-Lift Prediction Workshop
  - Full-span flap and slat configuration
  - Slat at 26.5°, Flap at 32°
  - Near-incompressible (Mach 0.175) flow conditions
Meshing in FlightStream:

- Mesh generated using FlightStream extruder and surface quad mesher (69.96% quad recovery)
- 8,236 surface faces
- 5,355 solver quads
DLR-F11

Solver run-time conditions:
- Parallel run conditions on 2 processors
- ~230 seconds run-time for single AOA condition till solver convergence
- Relaxed wake conditions on flaps and main wing
**DLR-F11**

**Results:**
- Angles of attack sweep matched to that in AIAA HLPW2 Experimental data
- Reynolds number: 15.1 Million
- Mach 0.175
Modeling Propellers In FlightStream

The Rotary Toolbox
The rotary toolbox in FlightStream is a wrapper around the core solver designed to create an easy-to-use environment for modeling propellers and wind-turbines.

The overall solver settings are similar to that for the steady-state models coupled with volume physics controls.

The rotary toolbox is also optimized to provide performance data relevant to propellers but which are less used for standard fixed-wing aircraft analysis.

The rotary toolbox has proven to be a powerful tool for very-fast analysis of propeller problems.
The FlightStream rotary toolbox has been validated against several of the propeller models available on the UIUC Database.

Some results are presented here for the validated propellers.

More information on the propeller database can be found here: http://aerospace.illinois.edu/m-selig/props/propDB.html

Propeller case-study: The Illinois propeller database

[Graph showing thrust coefficient vs. advance ratio for Super Nylon 9x5 Propeller]
FlightStream in use

Some applications
FlightStream has been used for analyzing the baseline geometry of the D8.5 aircraft concept in conjunction with NASA Langley.

A baseline model has been compared with Euler results at high compressible conditions over a range of different Mach numbers.

Results compare favorably and served as additional validation of the compressible fluid flow models used in FlightStream.
FlightStream has also been used for analyzing potential flap configurations on the D8.5 aircraft concept in conjunction with NASA Langley.

A flap configuration has been added to the baseline design and tested for different positions and deflections.

Ongoing work...
FlightStream in use: Missile roll-reversal

- FlightStream has been used for predicting roll reversal in missiles in conjunction with Army efforts at Huntsville.

- Geometry designed entirely in FlightStream by students at Auburn University using FlightStream CAD tools.

- Vorticity interference capture between canards and base-fins is captured accurately by the relaxed wake strand model in FlightStream; Overall run time is on the order of 90 seconds for each AOA.

- Results compare favorably with classical CFD approaches and are still being worked on as part of current efforts.
Questions?