vNIAR Virtual Engineering Laboratory

CFD Simulation and Wind Tunnel Test Correlation for a Tailless UAS

Luis Gomez | Gerardo Olivares Ph.D.

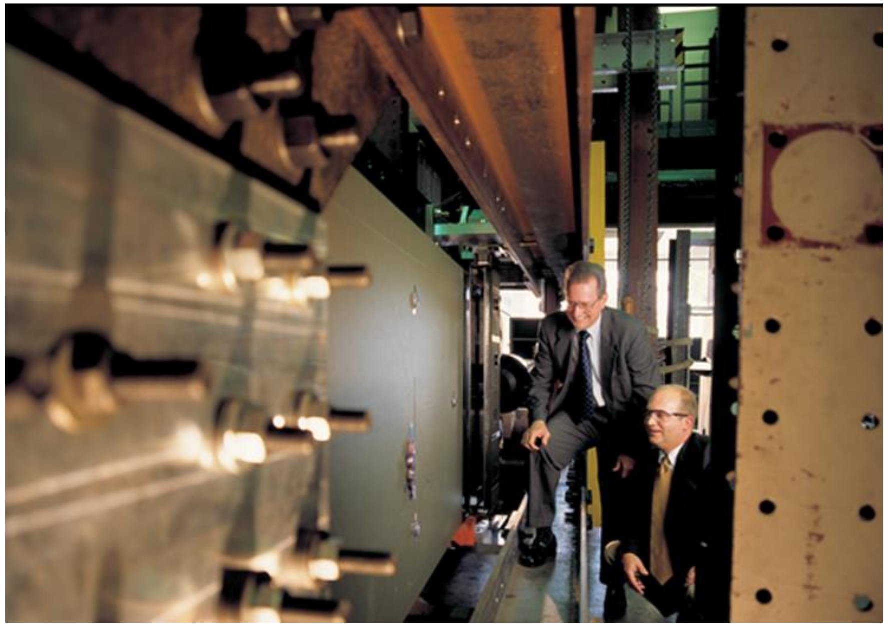
FloEFD Simulation Conference – Berlin, 29th November 2017

National Institute for Aviation Research

NIAR Mission

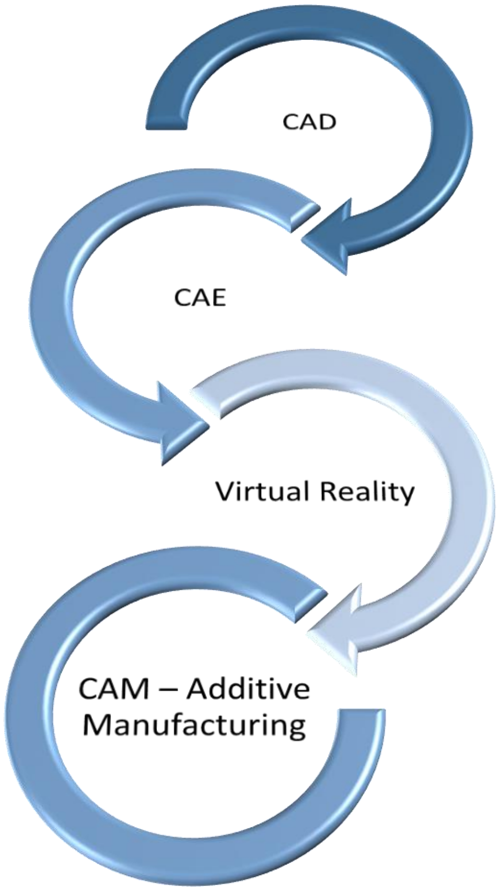
- The National Institute for Aviation Research (NIAR) at Wichita State University was established in 1985 in order to strengthen research and services support to the aviation industry.
- Located in Wichita, KS. "The Air Capital of the World"
- 150,000 square feet of laboratory space, NIAR is home to several major laboratories including: Virtual Engineering, Crash Dynamics, ATD Calibration, Advanced Joining, Aging Aircraft, CAD/CAM, Composites & Advanced Materials, Fatigue & Fracture, Full-Scale Structural Testing, Wind Tunnels, and Human Factors.
- Over 400 employees
- NIAR Provides:
  - research
  - design
  - certification testing
  - technology transfer
  - training

  to…
  - aerospace industry
  - aviation-related companies
  - non-aviation companies
  - federal aviation research sponsors

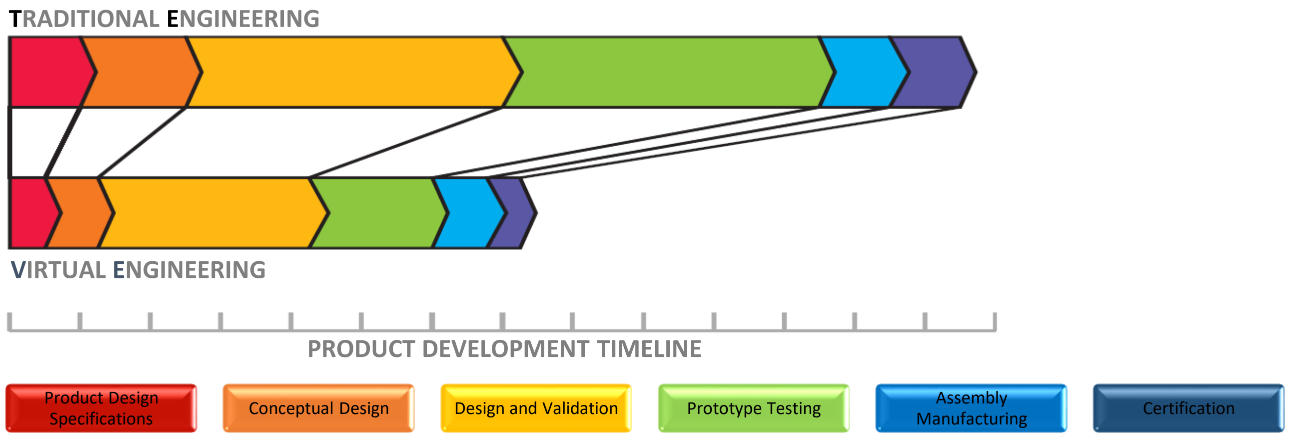


NIAR Virtual Engineering Laboratory

Virtual Engineering Group



- Virtual Engineering is defined as integrating geometric models and related engineering tools such as analysis, simulation, optimization, and decision making tools within a computer-generated environment that facilitates multidisciplinary collaborative product development.
- Advantages Virtual Engineering:
  - Minimize Physical Testing
  - Minimize Physical Product Revisions
  - Reduced Development Cycles
  - Reduced Certification Cycles
  - Improved Assembly and Manufacturing Cycles
  - Accomplish Results with a Less Experienced Work Force
  - Robust Design
  - Improved Product Knowledge
  - Innovation



Our Mission:
Develop a Search and Rescue sUAS

Multi-Mission/Multi-Variant

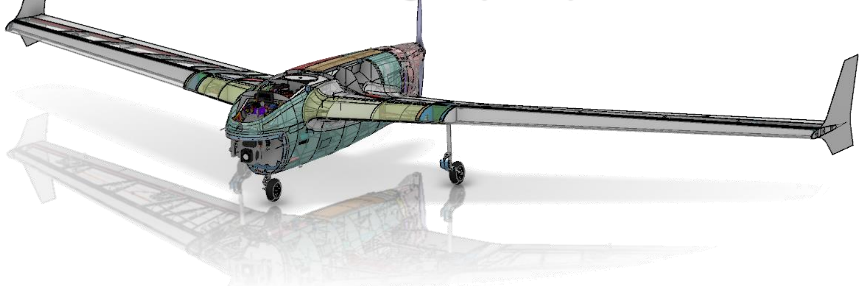
- 25 Kg weight
- 2.3 Kg payload
- 3.6 m span
- Hybrid propulsion
- C/V/HTOL
- Retractable landing gear

sUAS – Modularity

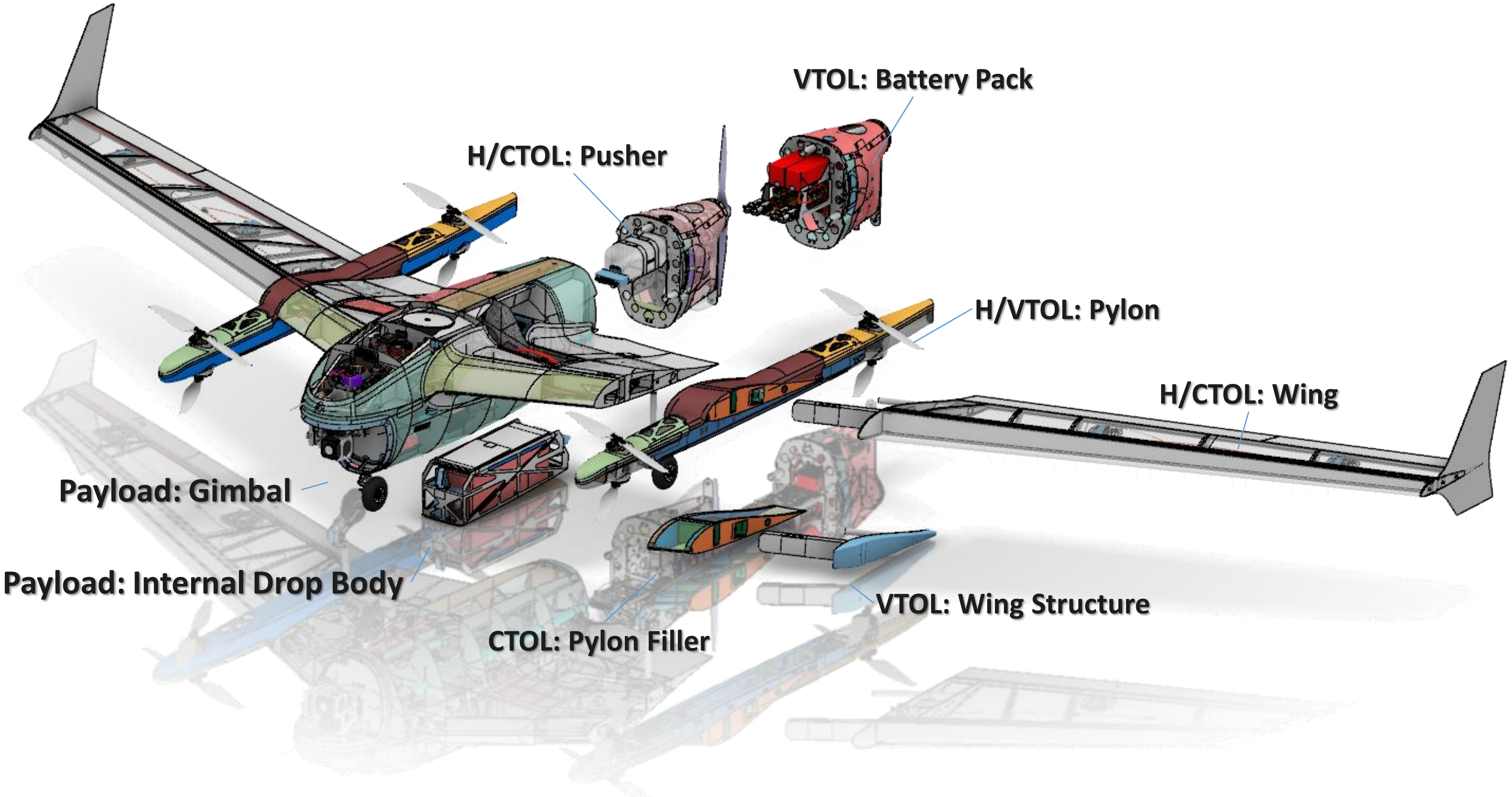
HTOL ([CTOL] + [VTOL])

Electric Engine (VTOL)

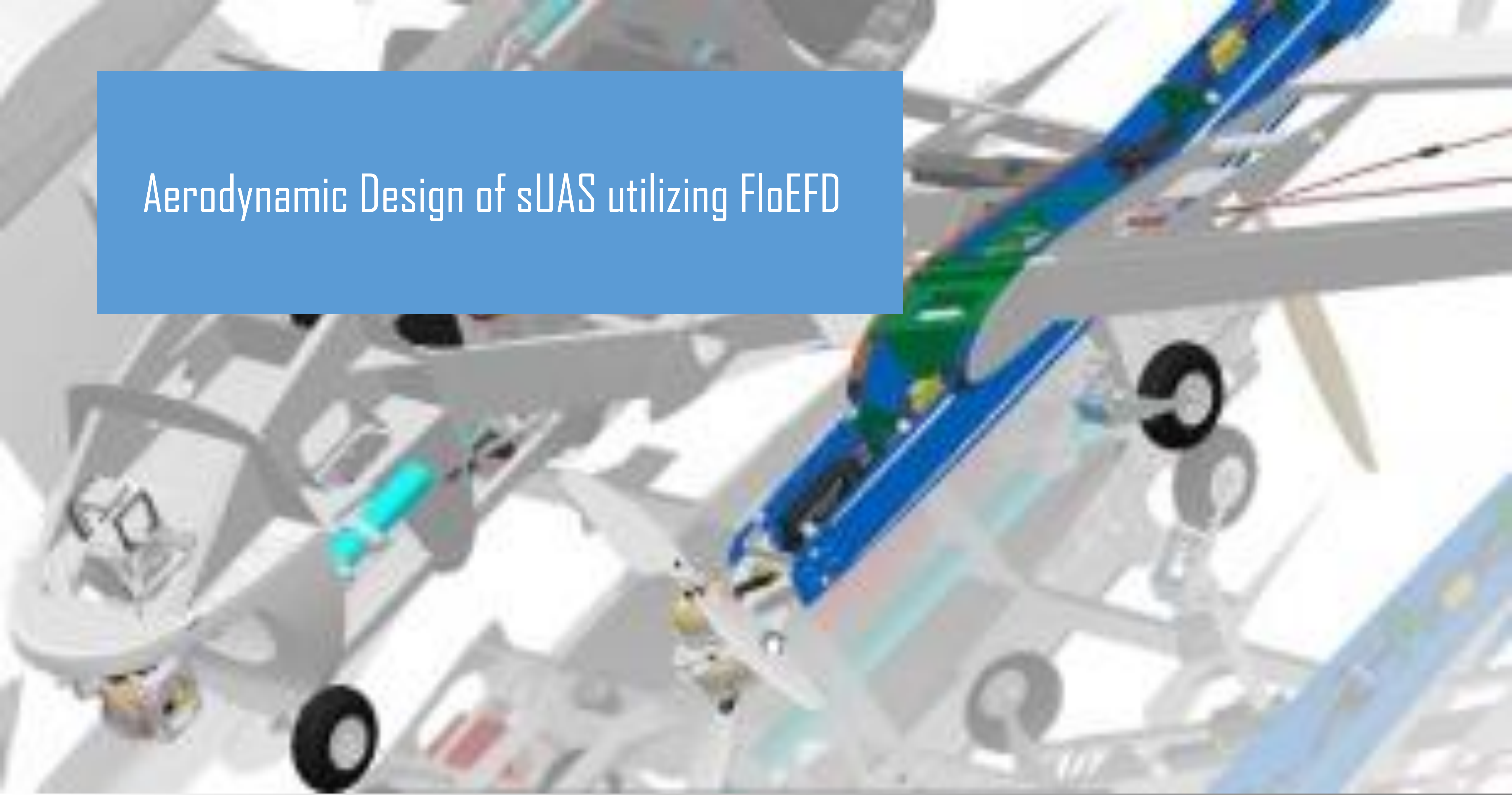
Piston Engine (CTOL)



Tailless Configuration, Gimbal system with zoom/infrared cameras



Aerodynamic Design of sUAS utilizing FloEFD



sUAS – Design Overview

- FloEFD is utilized to aerodynamically design the sUAS to meet the following stakeholder requirements:
  - Tailless, Pusher propeller configuration
  - Cruise velocity of 50 mph (22.35 m/s)
  - Maximum Take-Off Weight of 55 lbs (25 Kg)
  - Forward Flight Endurance of 5 hours
- Parametric study of the design for different combinations of airfoils, sweep & incidence angles, dihedral angles is carried out to optimize the aerodynamic performance and to achieve a stable configuration

sUAS Characteristics (HTOL):

Wing Span	3.58 m	C_L at Cruise	0.4
Aspect Ratio	7.56	Maximum C_L	1.15
Wing Area	1.68 m^2	Stall Speed	14 m/s
Airfoils	MH81/MH80	Stall Angle of Attack	10°
L/D Ratio at Cruise	13.5	Static Margin	5%

sUAS – CFD Analysis

Computational Model

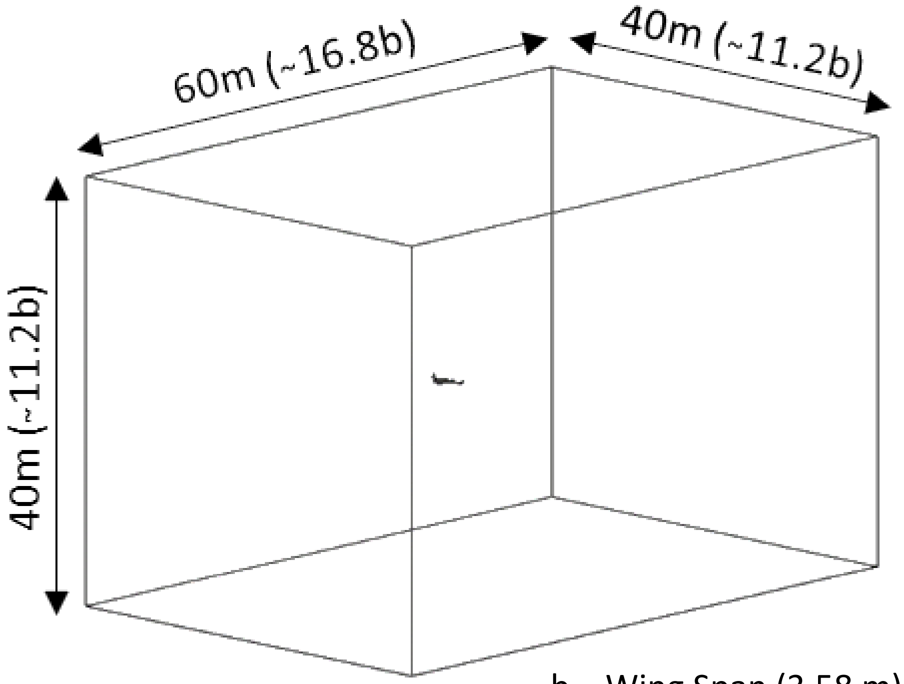
✓ **FloEFD Numerical Model:**

- Favre-Averaged Navier-Stokes equations, Finite Volume Discretization
- Low Compressible Solver: Pressure based, Implicit 2nd order accuracy of spatial derivatives, Implicit 1st order accuracy of time derivatives
- Modified *k-ε* two-equation turbulence model, Free Transition
- Cartesian Mesh

✓ **Two-Scales Wall Functions Model to resolve boundary layers:**

1. Thin Boundary Layer Method (proprietary Integral method technology)
   - Used when boundary layer thickness is more than the size of the first mesh element near the wall (coarse mesh)
   - Used in the analysis for low angles of attack and complete range of sideslip angles with Solution Adaptive Refinement (SAR)
2. Thick Boundary Layer Method (classical method)
   - Based on a Van Driest's velocity profile. Used when sufficient number of cells are present within the boundary layer (fine mesh)
   - Used in the analysis for high angles of attack

Rectangular Computational Domain



b – Wing Span (3.58 m)

sUAS – CFD Analysis

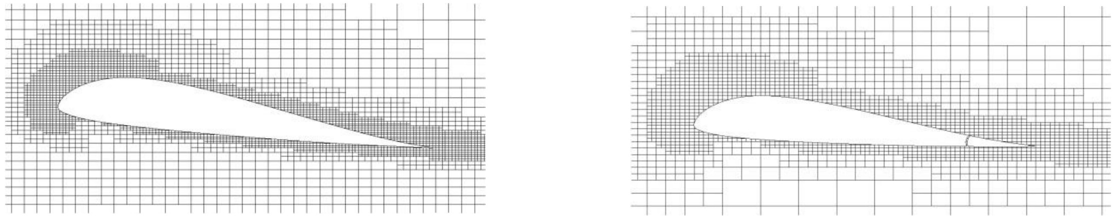
Thin vs. Thick Boundary Layer

- Thin boundary layer method is used for the conceptual design at low angles of attack and sideslip angles. Multiple configurations analyzed with a coarse mesh (lesser CPU time)
- Thick boundary layer method is used to improve the correlation with wind tunnel results at high angles of attack
- 4 to 5 cells (fine mesh) within the boundary layer thickness calculated analytically using Blasius method
- Mesh for 2 wing cross-sections of the final design for both methods is shown
- Comparison of the two methods:

Boundary Layer Type	Angle of Attack (degrees)	Initial Mesh Size (million cells)	Final Mesh Size (million cells)	Solution Adaptive Refinement	Typical CPU Time (hours)
Thin	0	≈ 0.35	≈ 4.5	Yes	≈ 11
Thick	10	≈ 13	≈ 13	No	≈ 54

Simulations performed on a 12 core Intel Xeon X5675 workstation

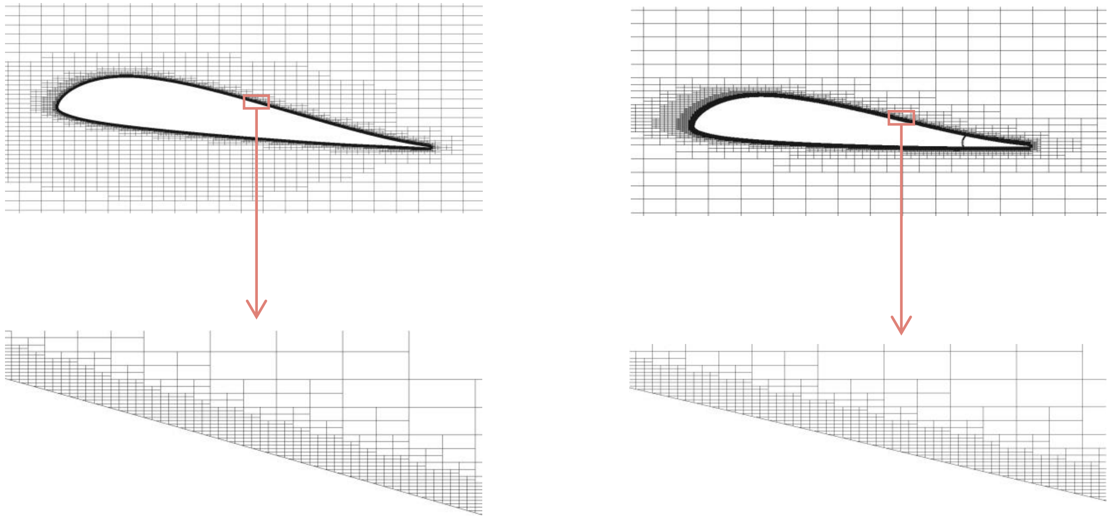
THIN BL method: Mesh Cut Plots (V = 22.35 m/s, α = 0°)



Near-Root Section

Near-Mid Section

THICK BL method: Mesh Cut Plots (V = 22.35 m/s, α = 0°)



Near-Root Section

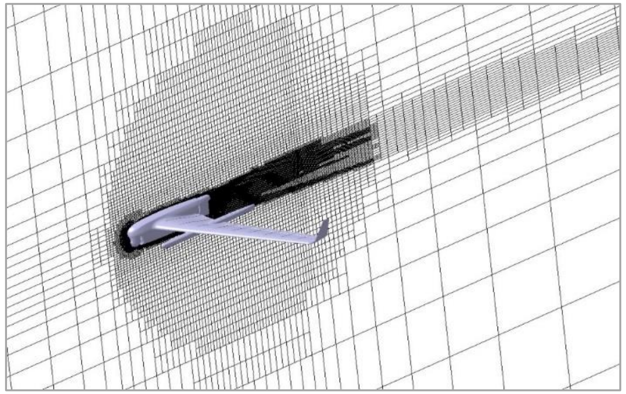
Near-Mid Section

sUAS – CFD Analysis

Grid Independence Study

- A grid independence study is performed for the thin boundary layer method
- Solution Adaptive Refinement (SAR) with 4 refinements is used for all analysis with the thin boundary layer method
- Results do not vary more than 7 % as the grid size is increased, therefore the solution satisfies grid convergence requirements. Mesh 2 which is the intermediate mesh (finer than Mesh 1 but coarser than Mesh 3) is used for all further analysis.

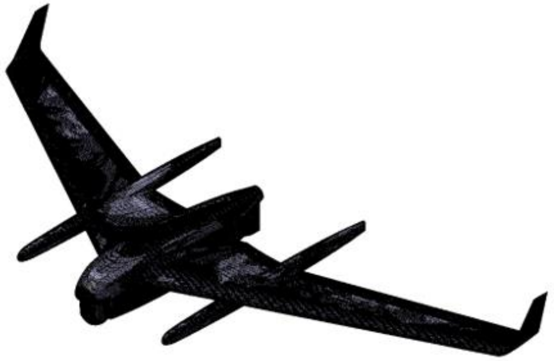
Mesh Contour with SAR (V = 22.35 m/s, α = 0º)



V = 22.35 m/s, α = 0°			
Case	**Mesh 1**	**Mesh 2**	**Mesh 3**
Final Mesh Size (million cells)	1.60	4.50	10.60
CPU time (hours)	4.80	11.46	35.10
C_L	0.4086	0.4060	0.4034
C_D	0.0304	0.0289	0.0296
C_M	-0.0041	-0.0041	-0.0039

Parameters	**Mesh 1 - 2**	**Mesh 2 - 3**
% change in C_L	1 %	1 %
% change in C_D	5 %	3 %
% change in C_M	2 %	6 %

Surface Mesh with SAR (V = 22.35 m/s, α = 0º)

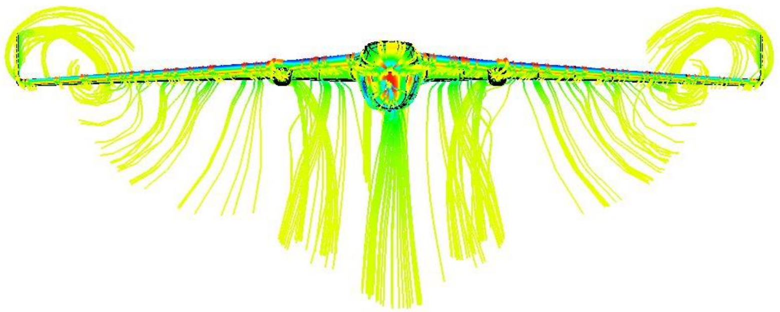


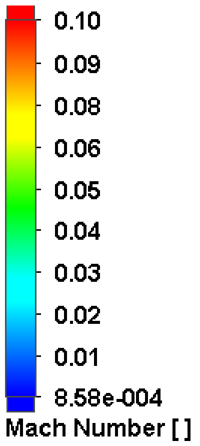
- For the thick boundary layer approach, the mesh is created such that there are 4-5 cells present within the boundary layer

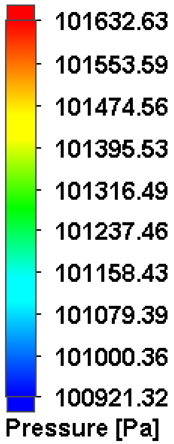
sUAS – CFD Analysis

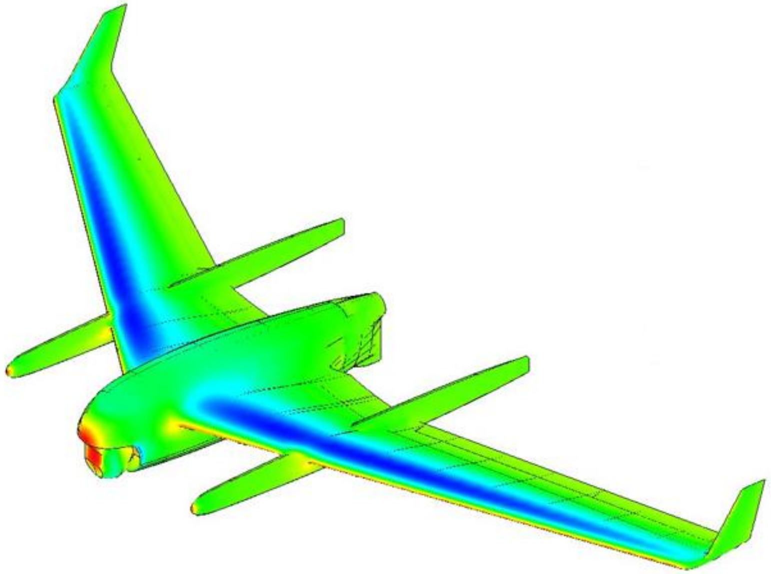
Surface & Contour plots: Thin Boundary Layer method

Flow Trajectories (V = 22.35 m/s, α = 0⁰)

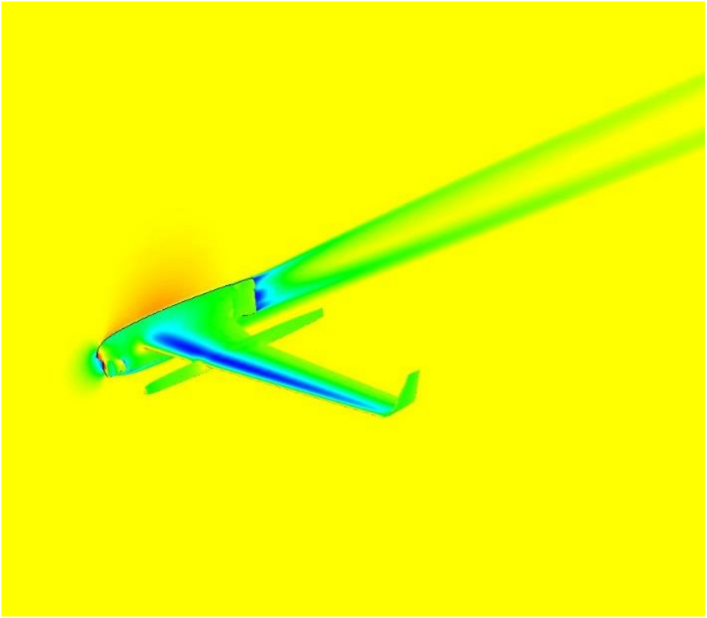








Surface Pressure Distribution (V = 22.35 m/s, α = 0⁰)



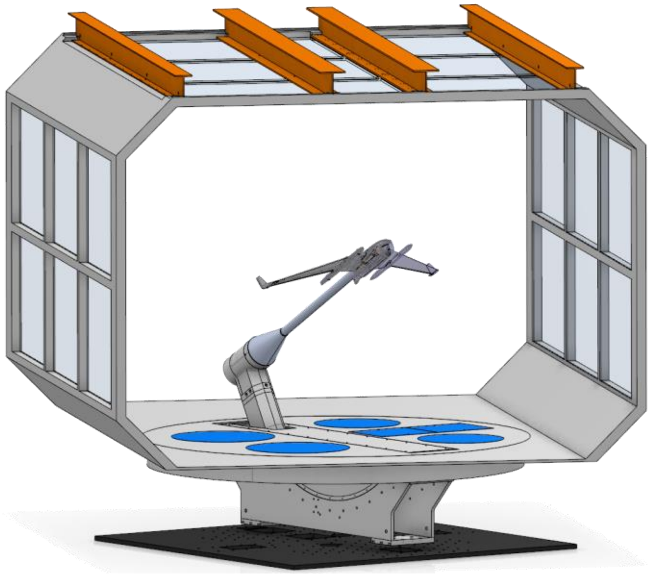
Mach Contour (V = 22.35 m/s, α = 0⁰)

sUAS Wind Tunnel Test

sUAS – Wind Tunnel Test

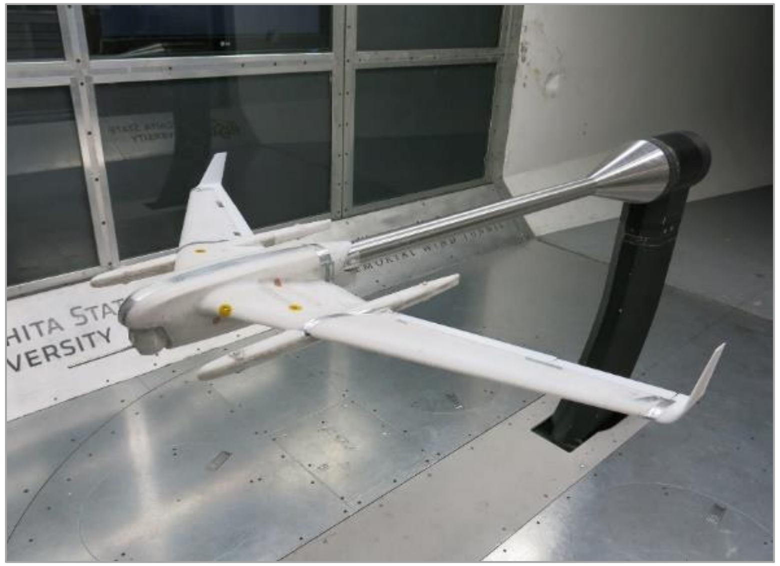
Test Details

- The NIAR Walter H. Beech Wind Tunnel is a subsonic, closed return and atmospheric type with test section of 7’x10’ in cross sectional dimensions
- The model configuration is a sting mount with internal balance (this configuration is used for large angles of attack)
- The model is printed at **<u>1/3 scale</u>** in order to fit within 3D printer limits



✓ **Test Matrix:**

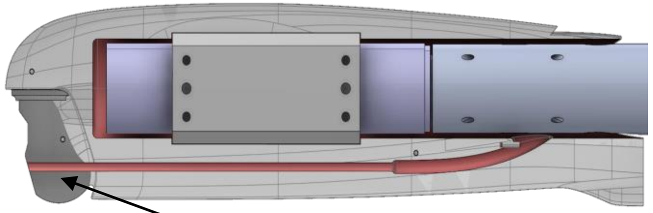
- Tunnel speed is $\approx$ 80 m/s to match a Reynolds No. of $\approx$ 750,000 at cruise
- Angle of attack sweep: -10° to 15°
- Sideslip angle sweep: -5° to 25°
- Elevon deflection sweep: -20° to 20°

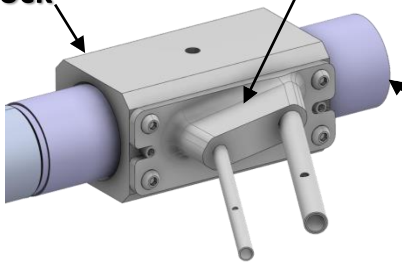


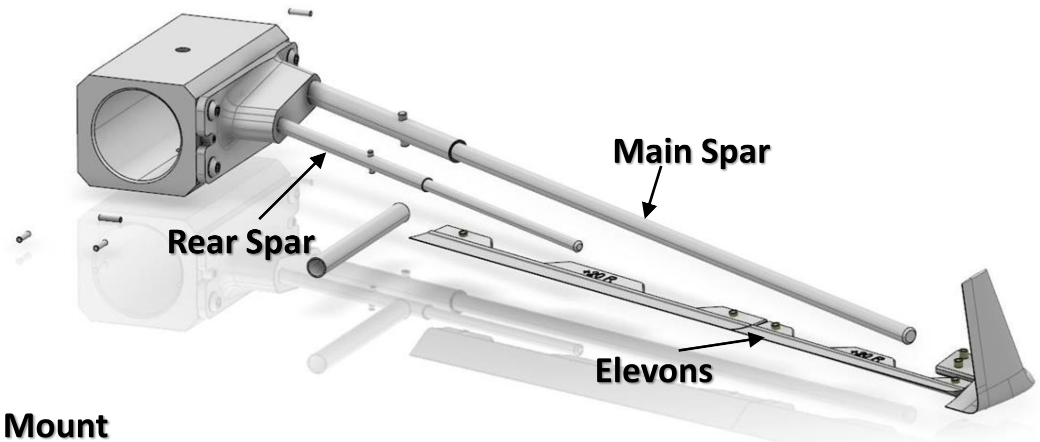
sUAS – Wind Tunnel Test

Model Overview

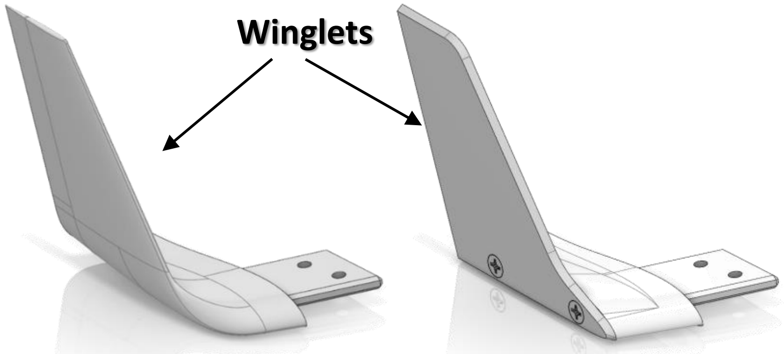
- The model is printed out of PC-ISO and hand-sanded to obtain a smooth finish
- The model has .010” clearance around the center block and is press fit on the block. The center block and adapter are made out of stainless steel. There is .075” clearance around the sting diameter.
- The internal structure comprises of the main spar and the rear spar made out of stainless steel tubes.







- The modularity of the model is created by the two bearing shafts being slid into the hollow tubes and then pinned with dowels
- The elevons are printed and then pinned for dimensional accuracy



- The winglet is replaceable with a flat plate simulant
- The part is also pinned to into the tunnel model

Verification & Validation of CFD Results

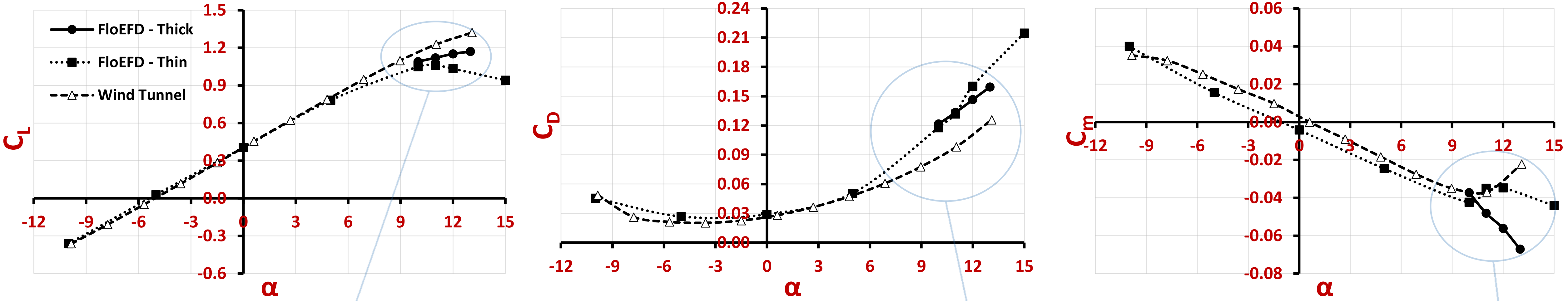
sUAS – Validation & Verification

Comparison of CFD & Wind Tunnel Results

✓ C_L & C_D prediction is in excellent agreement with wind tunnel results in the linear angle of attack range using **thin** boundary layer wall method

✓ C_m prediction is in good agreement with wind tunnel results (slight offset due to difference in geometry of wind tunnel model)

Comparison of Lift, Drag & Pitching Moment Coefficients (α = -10° to 15°, β = 0°)



✓ C_L prediction for separated flow is improved using **thick** boundary layer wall method

! C_D is over predicted at high angles of attack by both thick and thin boundary layer wall methods

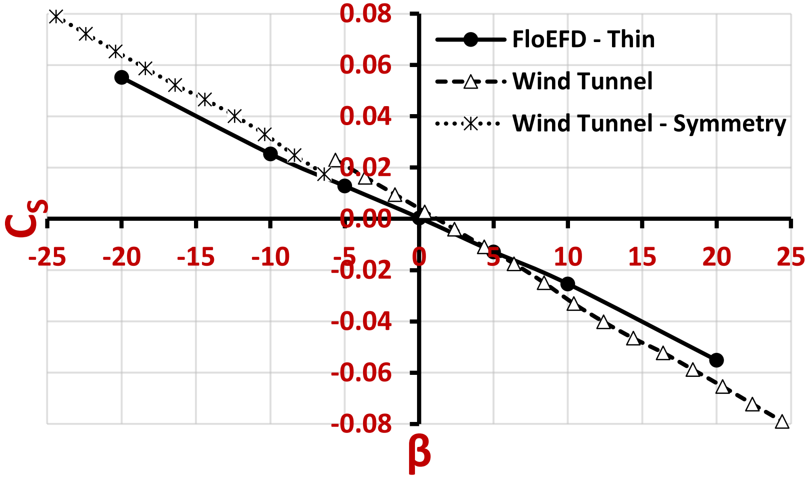
! C_M predicted for separated flow does not correlate with wind tunnel results

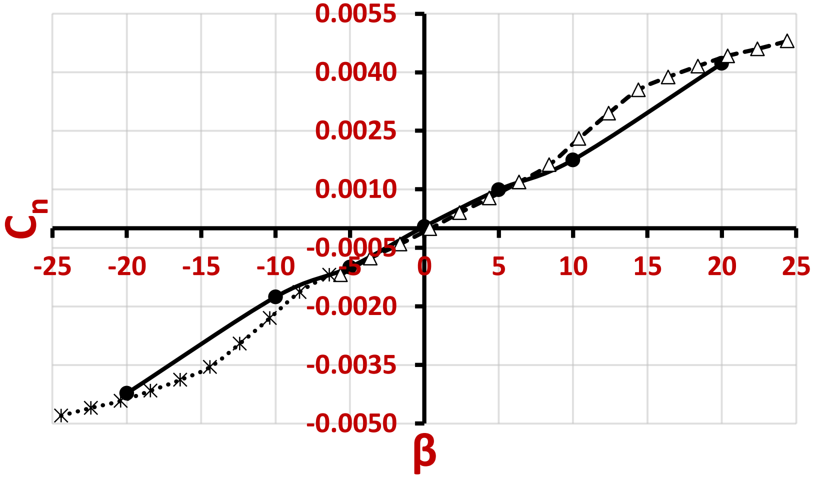
sUAS – Validation & Verification

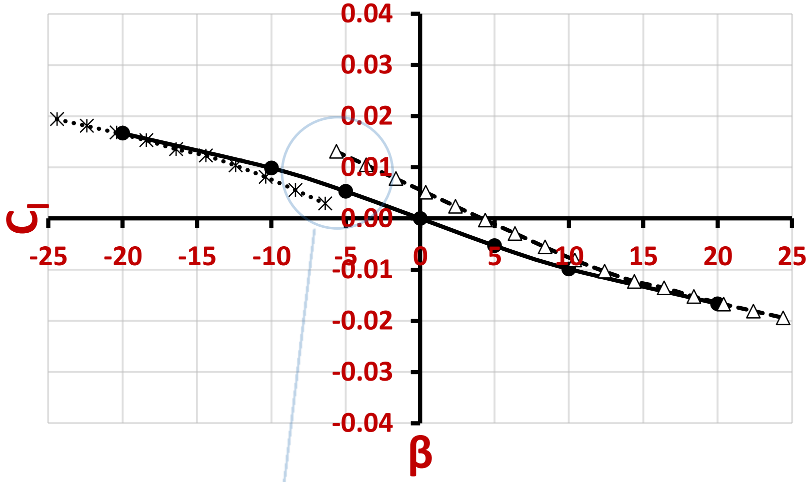
Comparison of CFD & Wind Tunnel Results (cont.)

- C_S (Side force coefficient) & C_n (Yawing moment coefficient) prediction is in good agreement with wind tunnel results using **thin** boundary layer wall method

Comparison of Side Force, Yawing Moment & Rolling Moment Coefficients (β = -25° to 25°, α = 0°)







Note:-

Wind Tunnel tests were carried out for β = -5° to 25°. Symmetry is assumed for results at all other values of β.

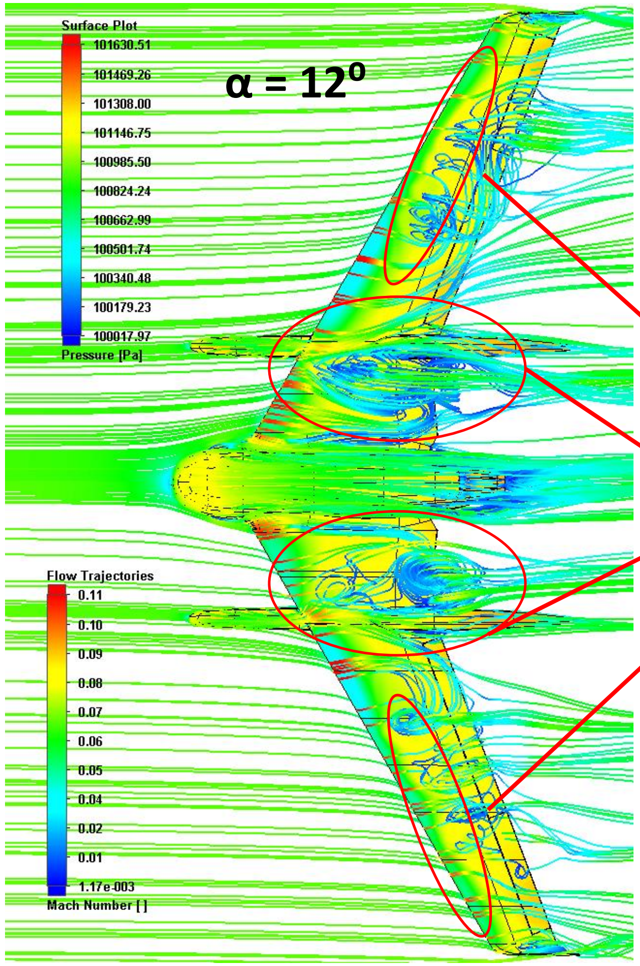
C_l (Rolling moment coefficient) from wind tunnel data is not symmetric

Investigation for this behavior is being carried out (could be a bad test data point)

sUAS – Validation & Verification

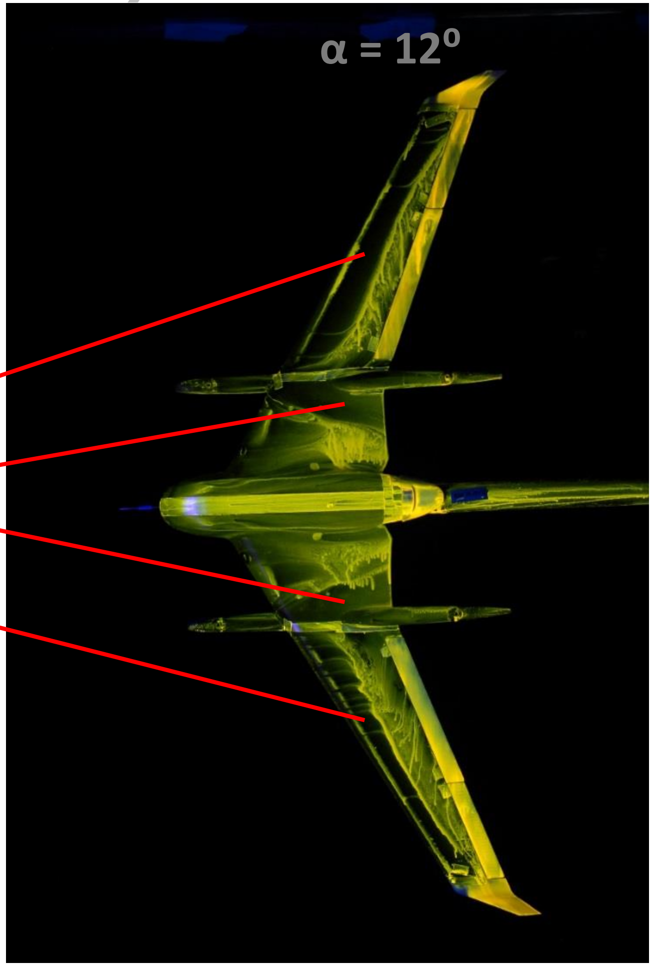
Comparison of CFD & Wind Tunnel Results (cont.)

FloEFD accurately predicts location of flow separation at high angles of attack as evident from wind tunnel flow visualization



CFD Flow Trajectories

Regions of Separated Flow



Wind Tunnel Flow Visualization

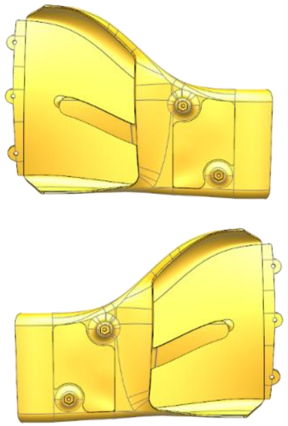
Thermal Management of sUAS Piston Engine

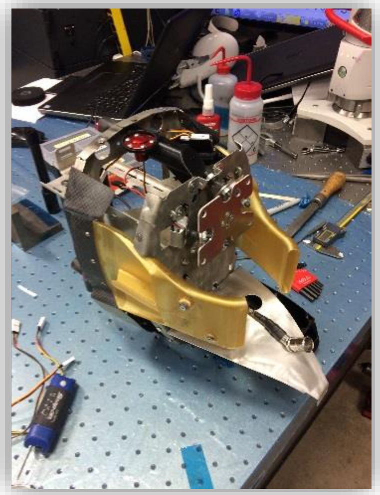
sUAS – Piston Engine Cooling

Problem Description

- Lab tests of the engine indicated high operating temperatures of critical engine components
- There was need for additional cooling as the pusher configuration provides very little active cooling
- Constraints:
    - Liquid cooling systems & Electric blower/suction fans add complexity and weight
    - Design OML frozen and cannot be modified

- **Solution:** Design ducts to provide additional cooling
- Ducts are 3D printed from Ultem 1010 and weigh < 200 g





DLE-35RA Piston Engine

- **Displacement:** 34.9 cc, **Bore:** 38.5 mm, **Stroke:** 30 mm
- **RPM Range:** 1,500-8,500, **Power:** 4.1 HP @ 8,500 rpm
- **Fuel:** Unleaded gasoline (minimum of 87 Octane)

sUAS – Piston Engine Cooling

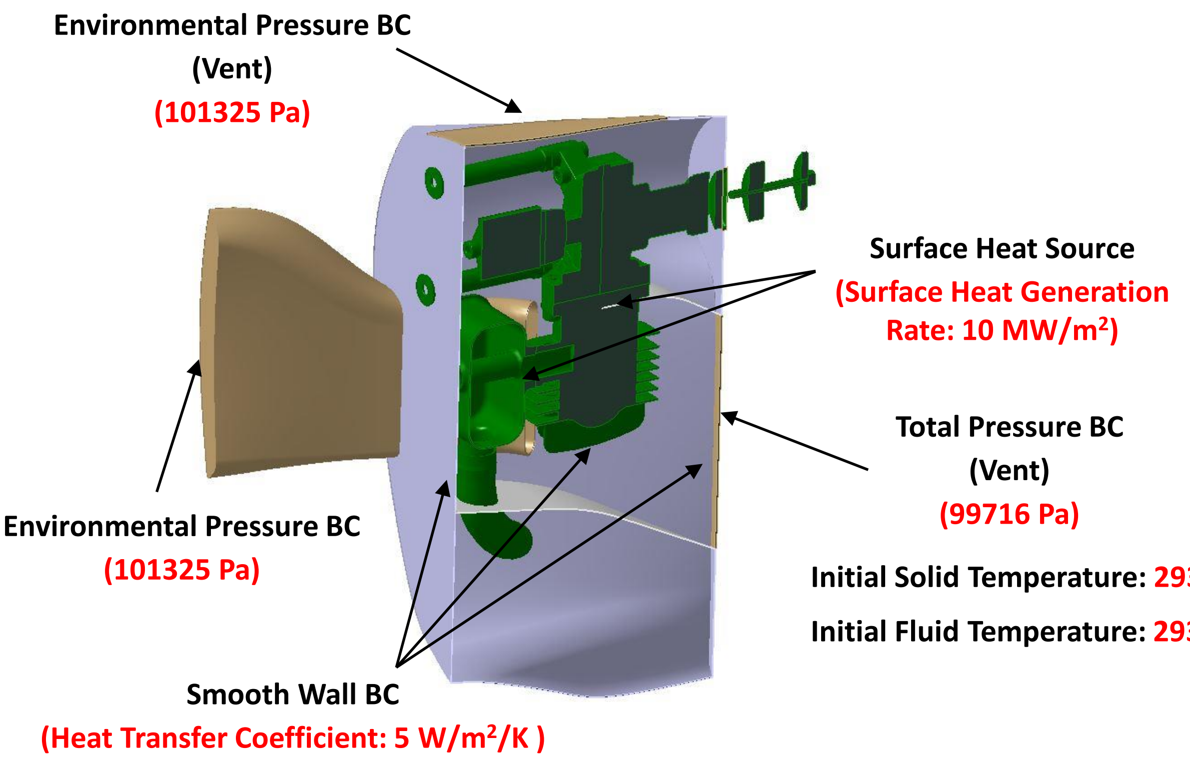
Conjugate Heat Transfer (CHT) analysis utilizing FloEFD

- Conjugate Heat Transfer Method: Heat Transfer (Fourier's Law, Newton's Law of Cooling) due to conduction in solids + Heat Transfer due to convection in fluids (Navier-Stokes equations)
- CHT Analysis used as a qualitative tool to determine the performance of the ducts and vent openings
- Multiple configurations analyzed. Results for the best design are presented here

✓ **Numerical Model Details:**

- Thin Boundary layer
- No. of Mesh Elements: ≈ 3 million
- CPU Time: ≈ 10 hours (12-core Intel Xeon X5675)
- Surface Heat Generation Rate & Heat Transfer Coefficients are approximate values from literature

Boundary Conditions

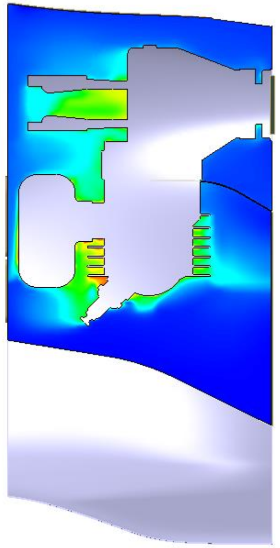


Initial Solid Temperature: 293 K

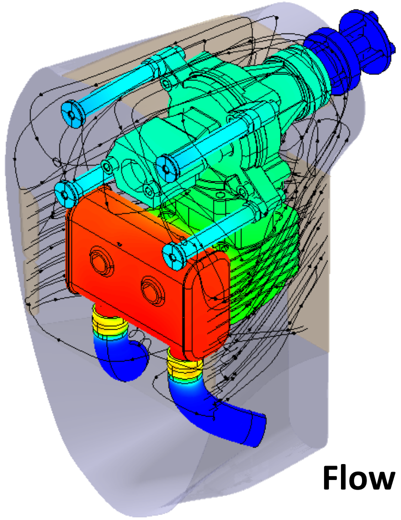
Initial Fluid Temperature: 293 K

sUAS – Piston Engine Cooling

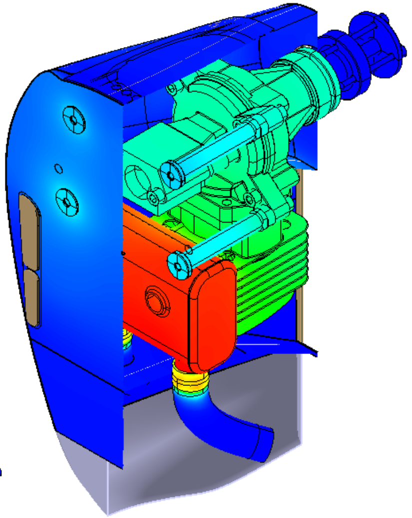
CHT Analysis Results: with & without Ducts



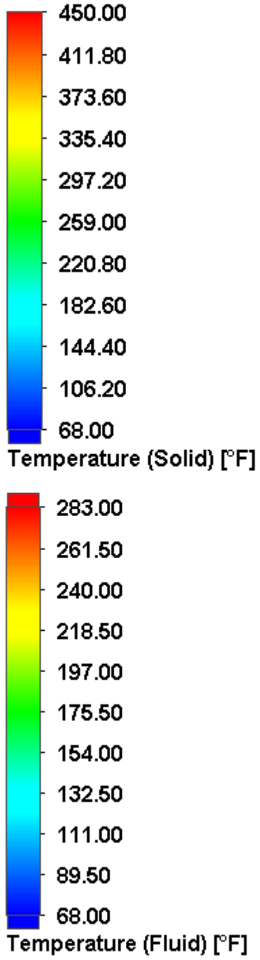
Fluid Temperature

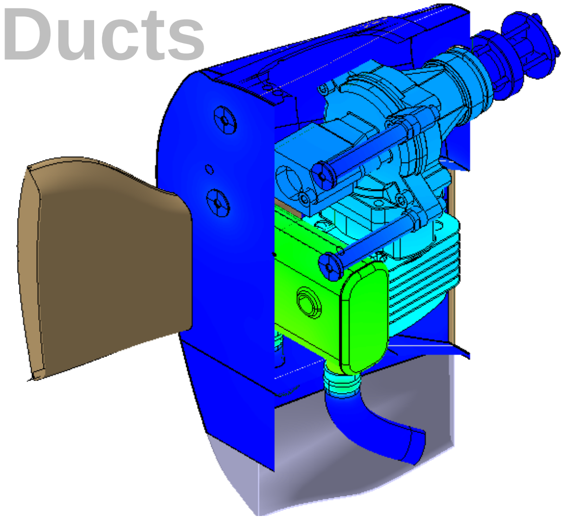


Flow Trajectories

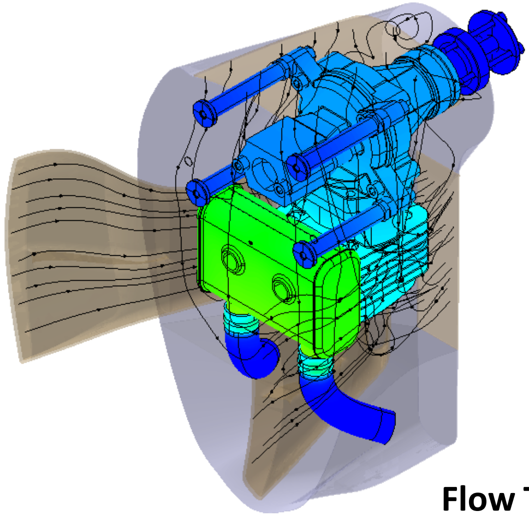


Surface Temperature Distribution – Skin & Engine

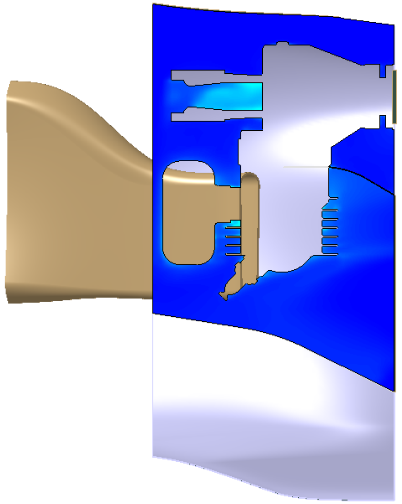




Surface Temperature Distribution – Skin & Engine



Flow Trajectories



Fluid Temperature

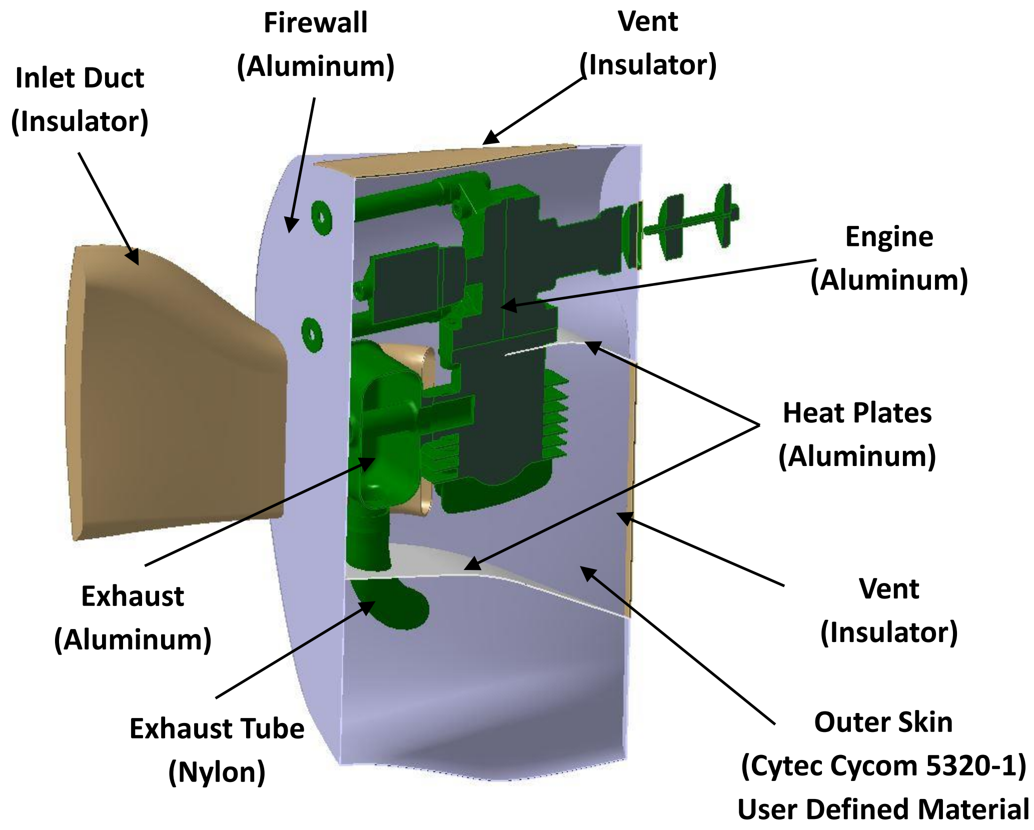
sUAS – Piston Engine Cooling

Conclusions & Remarks

✓ A reduction of ≈ 20% in maximum temperature of the engine is obtained with ducts and vents incorporated in the model

- A qualitative analysis to determine the improvement in cooling performance of ducts and vents is performed
- Materials available in the FloEFD database are used to define most of the components of the engine assembly
- Outer skin which is made of Cytec Cycom 5320-1 is modeled as a user defined material. Values of thermal conductivity and specific heat are based on literature data of similar materials
- Analysis is conservative since the pressure boundary condition used corresponds to the idle speed of propeller (1500 rpm)

Materials Definition

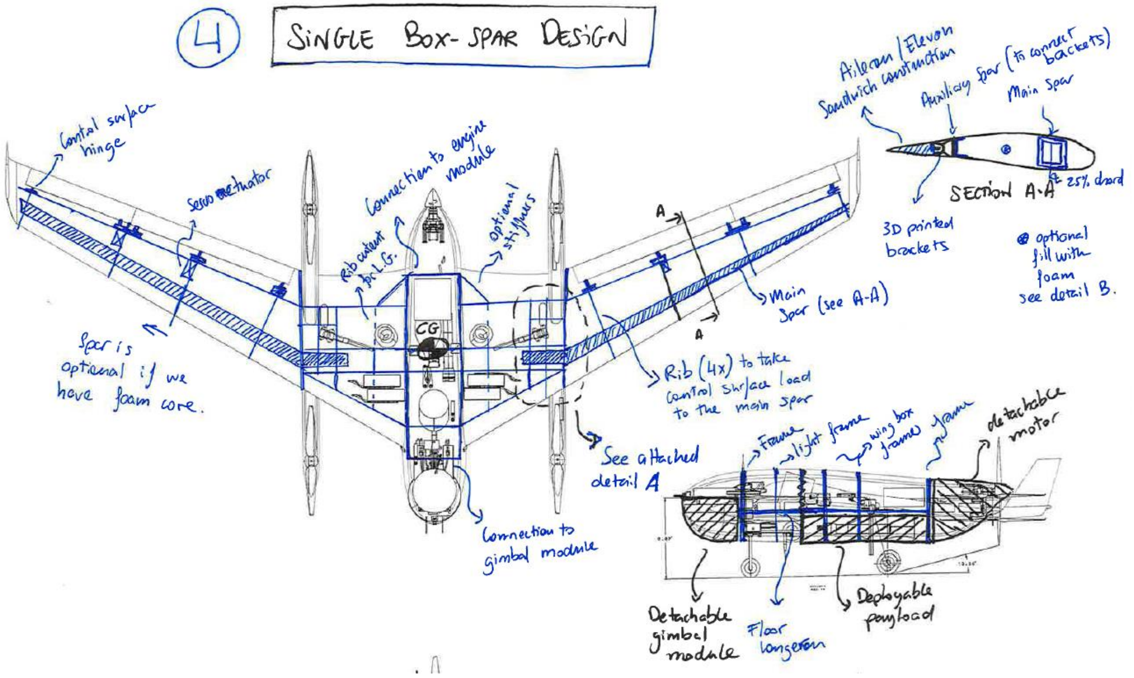


Structural Design of sUAS

sUAS – Structural Design

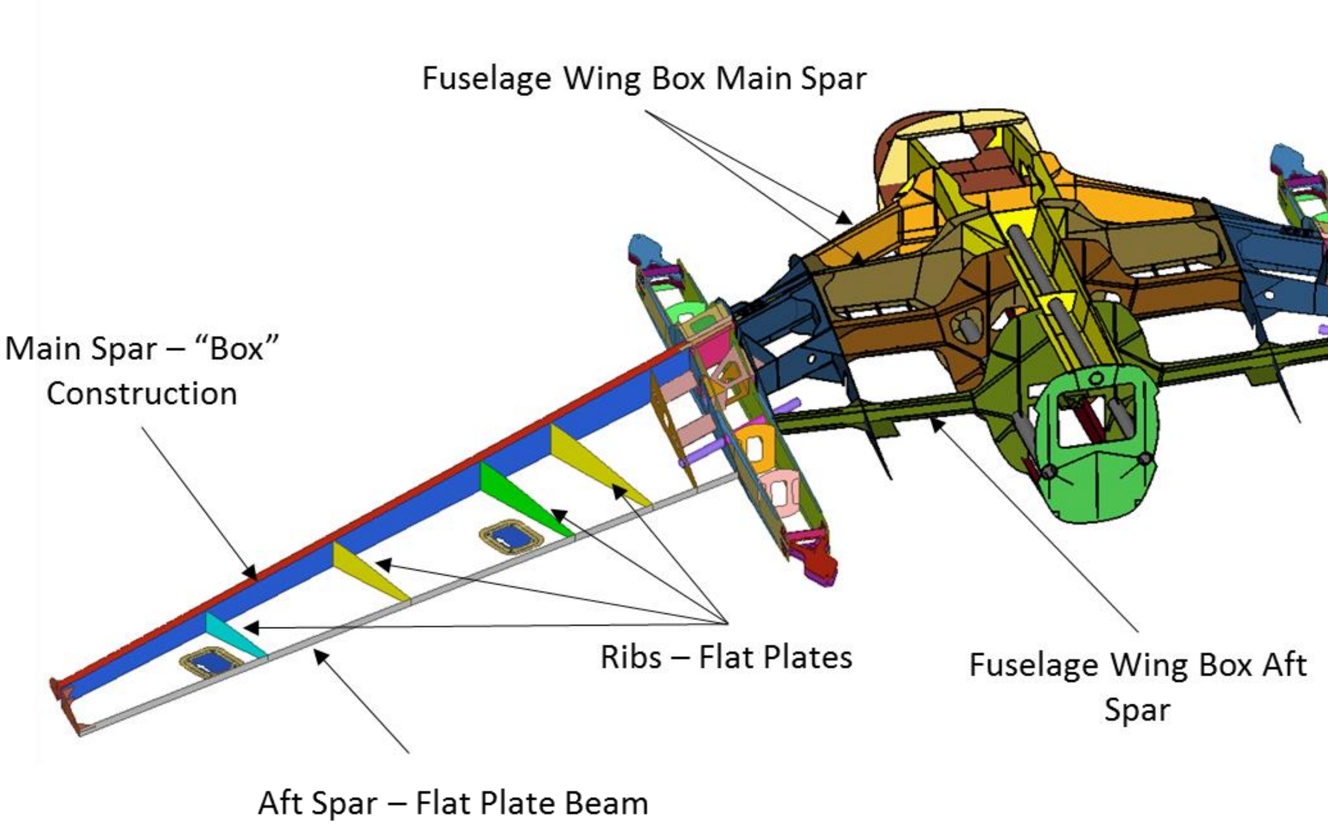
Concept Overview – Structural Components

Conceptual Design Hand Sketch



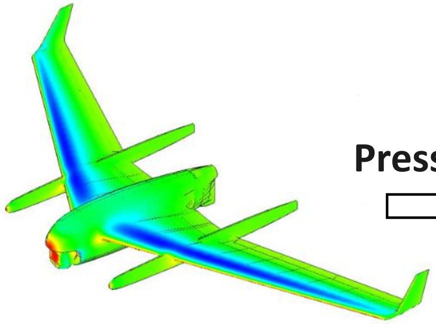
Outer Skin + Internal Structure:
>90 % Carbon Fiber (Cytec Cycom 5320)

Final Structural Layout

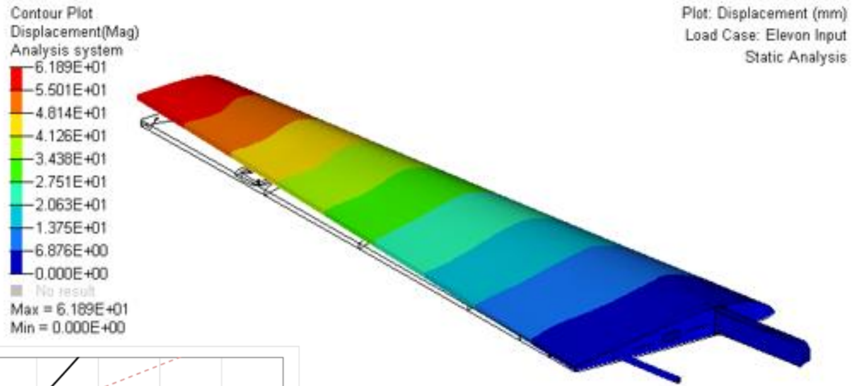


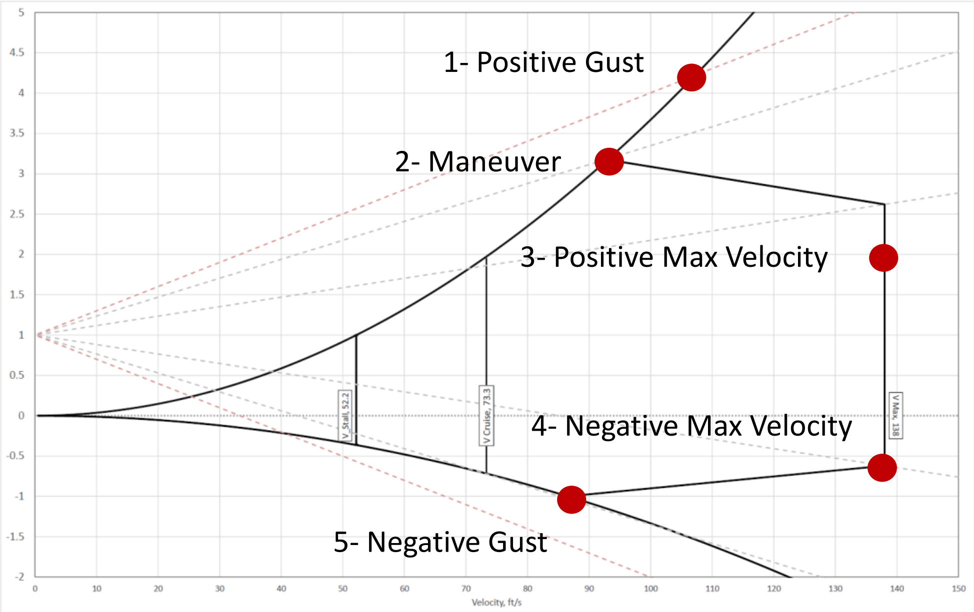
sUAS – Structural Design

Loads Mapping: CFD Model to Outer Wing FE Model



Pressure Loads





Loads mapping Procedure

- CFD analysis is carried out to get pressure loads at conditions shown in the V-n diagram
- Pressure distribution is exported from FloEFD as a text file for each load case
- It is mapped to the FE model using standard interpolation methods

FE Model Overview

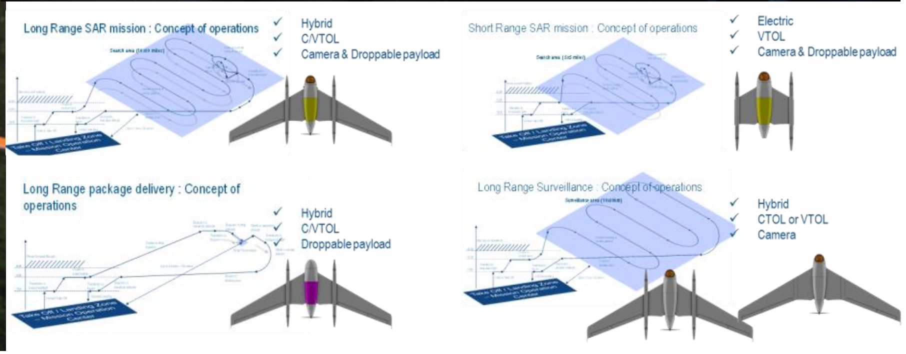
- Geometry idealization (from actual CAD model)
  - Thin surfaces simplified to 2D at mid-surface
  - Joints were idealized as a perfect connection of edges
  - Fasteners were modelled with 1D
  - Control surfaces and wingtip were only considered as non-structural point masses
- Mesh
  - 1st order quad elements (255,000 count)
  - Average mesh size of 1.5 mm
  - Quality control as per NIAR standards
- Materials
  - CFRP Weave - Cytec Cycom 5320-1 T650 3k-PW
  - CFRP-SM Tube - Rockwest 35137
- Properties
  - Composites were modelled considering a laminate of up to 4 plies.
  - Principal direction aligned with ¼ chord

vVirtual Flight Testing Laboratory

vNIAR

Virtual Flight Testing

Mission Analysis using Virtual Reality



NIAR