OpenVSP Structural Modeling Capability

Presented by:
Justin Gravett
Empirical Systems Aerospace, Inc.

Work Previously Supported by:
AFRL SBIR PHASE II: FA8650-15-C-2570
Agenda

• Introduction
• Development Overview
• Structural Entities
• VSP Terminology
• Typical Workflow
• GUI Overview
  – FEA Part Types
• Example Analysis
• CAD Structure Exports
• Demo
Introduction

• Objective:
  – Support modeling of aerospace structures & FEA mesh generation in OpenVSP

• Features:
  – Parametric modeling of full depth structural members
  – Extension of sub-surface capabilities to structural modeling
  – Support for beam & shell FEA elements
  – Key point identification of FEA nodes
  – Export to NASTRAN, Abaqus/Calculix, & Gmsh FEA mesh formats
Development Overview

Structures v2 Capabilities:
- Maximum one structure per vehicle
- Only wing geometries with ribs & spars
- Shell elements only
- No support for material & property assignments
- Point mass not included in mesh

Structures v3 Capabilities:
- Multiple structures on multiple geometries
- 12 total FEA part types
- Shell & beam elements
- Materials & properties library
- Mesh visualization tools
- Point mass as fixed mesh node
Structural Entities

Full-depth:
- Shell type of structure
- Extend completely through skin
- Examples: rib, spar, floor, pressure dome, fully-closed bulkhead

Zero-depth:
- Beam type of structure
- Sub-surface edge or intersection curve
- Cross-section properties
- Examples: stringer, longeron, stiffener

Key-points:
- Fixed FEA node
- Provide connectivity to other structures
- Application of point load or mass at specific locations

Partial-depth:
- Shell or beam types
- Do not extend completely through skin
- Examples: former, frame, partially-open bulkhead
- NOT SUPPORTED

Reference:
https://www.faa.gov/regulations_policies/handbooks_manuals/aircraft/amt_airframe_handbook/media/ama_ch01.pdf
VSP Terminology

Structure
- A collection of FEA Parts
- More than one structure allowed per geometry
- Only one geometry per structure

FEA Part
- A structural entity
  - Includes full-depth, zero-depth, key-points, etc.
  - FEA Element: A group of interconnected FEA Nodes
    - Assigned element properties
  - FEA Node: A coordinate point used to build FEA Elements
    - FEA Nodes can be assigned to multiple FEA Elements
    - Fixed Point is a specific FEA Node
  - Shell: Triangular FEA elements (full-depth)
    - NASTRAN: CTRIA6; Abaqus/Calculix: S6
  - Cap: Beam FEA elements (zero-depth)
    - NASTRAN: CBEAM; Abaqus/Calculix: B32
Typical Workflow

1. Add Geom
2. Analysis -> FEA Mesh...
3. Add Structure
4. Add and Modify Parts
5. Run “Mesh and Export”
6. Optional Settings
   - Add Materials
   - Add/Modify Properties
   - Adjust Mesh Settings
   - Specify Output Files
7. External Tools
   - Calculix
   - NASTRAN
   - Abaqus
   - Gmsh
8. Post-Process and Analyze
9. Visualize Results
GUI Overview: Structure Tab

**Features:**
- Structures for all geometry types except Blank & Hinge
- Multiple structures per geometry
GUI Overview: Part Tab

Features:

• Selection & control of multiple FEA parts
• Reordering for sub-surface overlap priority
• Shell: triangle elements
• Cap: beam elements
1. Slice
   - Cutting plane defined by orientation, center location, & rotation
   - Orientation: parallel plane for slice defined from body or absolute axes
   - Distance specified as relative or absolute
     - Relative: fraction of length
     - Absolute: fixed location
   - Slice can be full-depth, zero-depth, or both

![FEA Part: Slice](image)
2. Rib (wing only)
   - Slice defined from leading edge to trailing edge
   - Option for perpendicularity to wing leading edge, trailing edge, or any spar

3. Spar (wing only)
   - Slice defined from root to tip
   - Option to constrain to specific wing section
FEA Part: Dome

4. Dome
   - Semi-ellipse cutting surface
   - Defined by radius (A, B, C), center location (X, Y, Z), & rotation
   - Note: Must be oversized correctly to intersect skin
   - Not available for Wing geometries
5. Fixed Point
   • Forced FEA Node
   • Specified on parent skin surface or any FEA Part surface
   • Defined in surface coordinates (U, W)
   • Point mass support
FEA Part: Sub-Surface

6. Line
7. Rectangle
8. Ellipse
9. Control Surface

FEA Sub-Surfaces:
- All typical sub-surface options & parameters available for FEA sub-surfaces
- Tag: Identifies triangular shell elements for the sub-surface
- Cap (no tris): Only beam elements, used to create holes or stiffeners
FEA Part: Array

10. Rib Array
11. Slice Array
12. Line Array
   • No shell elements

FEA Arrays:
- Group of FEA Parts defined by spacing, starting location, & direction
- Distance: “Relative” for fixed # of parts; “Absolute” for fixed distance between parts
- “Individualize” creates independent FEA Parts
  - Note: This action cannot be reversed
GUI Overview: Material Tab

Features:
- Only linear, temperature-independent, and isotropic materials
- Material library available to all structures
GUI Overview: Property Tab

Features:
- Property library available to all structures
- All properties assigned a material
- Properties specific to element type
- Shell property: Set thickness
- Beam property:
  - General: Set cross-section area & area moment/products of inertia
  - Circle, Pipe, I, Rectangle, & Box: Set dimensions directly
GUI Overview: Mesh Tab

Features:
- Options identical to CFD Mesh
- Mesh settings specific to each structure
- Half mesh for symmetric structures
GUI Overview: Mesh Tab

Features:

- Mesh Exports:
  - `.stl` and `.msh` (no material properties)
  - Mass data file
  - NASTRAN
  - Abaqus/Calculix

- Surface & Intersection Curves:
  - Identical to Surface Intersection and CFD Mesh
GUI Overview: Display Tab

Features:
- Colors distinguish elements and nodes for different FEA Parts
- Visualization of triangle orientation vector and beam normal vector
- Separate sets for shell and beam elements
Example Analysis

- **OpenVSP FEA Mesh**
- **Set Boundary Condition Nodes**
- **Export to Calculix**
- **Set Load Nodes**
Example Analysis

Concentrated Load on Wing Tip Nodes

Modal Analysis: Mode 2 Displacement
CAD Structure Export

- CAD export of structural models
  - STEP or IGES formats
  - Analytical NURBS surfaces
  - Unintersected (not trimmed)
- Enables external FEA mesh generation tools
- Planned development of trimmed CAD exports
Demo
Contact Information

Justin Gravett
justin.gravett@esaero.com

ESAero
openvsp@esaero.com
Support Slides
API & Scripting

• Structure Functions in API:
  – AddFeaStruct( const string & in geom_id, bool init_skin = true, int surfindex = 0 )
  – DeleteFeaStruct( const string & in geom_id, int fea_struct_ind )
  – SetFeaMeshVal( const string & in geom_id, int fea_struct_ind, int type, double val )
  – SetFeaMeshFileName( const string & in geom_id, int fea_struct_id, int file_type, const string & in file_name )
  – ComputeFeaMesh( const string & in geom_id, int fea_struct_ind, int file_type )
  – AddFeaPart( const string & in geom_id, int fea_struct_id, int type )
  – AddFeaSubSurf( const string & in geom_id, int fea_struct_id, int type )
  – AddFeaMaterial( )
  – AddFeaProperty( int property_type = 0 )

• Examples:
  – \repo\src\geom_api\APITestSuite.cpp
  – \repo\examples\scripts\FEAMesh.vspscript