Suggestions for Finite-Element Analyses of AAE450 Structures
by Stephanie Van Y, 450 TA, Fall 2003

Structural Design

1) Structural Representation

Outlined here are some basic requirements a structural model must meet. This discussion is in no way meant to cover all criteria that must be met by the vehicle designed. The structural representation of your vehicle must be appropriate for the vehicle size being designed. This means that if the load bearing structure of your vehicle is to be contained within an outer mold line, than it must fit within that outer mold line of the vehicle. Along with this, any cargo that would naturally be contained within the load bearing structure of the vehicle must also fit within the structural representation. Finally, the load bearing structure of your vehicle must be capable of withstanding the critical loading conditions for the designated [2].

The structural designer should do initial investigation into the best type of structure which might be appropriate for the general vehicle shape decided upon. The most logical place to find this type of information is to look at previous vehicles that were designed for a similar mission. Web resources as well as previous design reports are a good starting point. If this does not provide enough information then consulting a professor or professional who would have insight into the type of structure that would best suit the type of vehicle (size and shape) your team has decided upon.

Be aware that your vehicle size and perhaps shape may change over the course of the semester design cycle. Once an initial structural design is chosen determining the versatility of this design would be a feasible design trade initially. What is the maximum/minimum feasible volume a particular design shape can contain? Understanding the complexity of this design with regards to the analysis to be conducted should also be considered. If the shape is so complex that it cannot be modeled within the semester design time then an alternative design must be considered.

2) Determine Critical Structural Loading Conditions for Mission

The team shall decide upon the mission to which the structure will be designed. This does not require that every leg of the mission be established before structural design can begin.

Historically, with regards to the aero-assist vehicle mission, two loading cases have been identified as the critical loading conditions:

a) launch loads
b) parafoil deployment

These critical loading conditions will be adjusted according to the specific mission to be accomplished.

3) Conduct Design Analysis for Structural Critical Loading Conditions

As discussed in the Methods Handout [2] the structural designer must design your vehicle’s structure to accommodate the most critical loading conditions experience by the vehicle throughout the mission. The critical loading conditions for a typical aero-assist vehicle have been identified as the launch conditions and the parafoil deployment. These loading conditions correspond to vehicle buckling and bending constraints. Buckling is the most obvious for the launch sequence while bending would be predominant in the parafoil deployment (presuming the parafoil over the c.g.).

The following outline is to be used as a guide to conducting a bending and buckling analysis using the ANSYS software available. If the structural designer is not familiar with ANSYS it is recommended that they work through some of the online tutorials to become familiar with the functionality of the software. It is recommended that new users begin with the Structural Tutorial (designing an L-bracket).

Analysis Tools

This senior design course does not allow enough time for a full model finite element representation to be analyzed.

The Finite Element Code recommended for analysis on your vehicles is Ansys. You can access Ansys from any Purdue ECN machine.

Start → ECN Software → ANSYS 7.0 → ANSYS

A Window should pop up with a large black area and a coordinate system in the middle. Under the Help menu at the top of the window find ANSYS Tutorials.

It is recommended that the students follow some of the tutorial examples provided in the Ansys Tutorials. The first structural tutorial is the ‘bracket’. This tutorial provides a quick introduction to creating geometry in Ansys as well as conducting analysis.

The Ansys help ‘search menu’ can be useful in understanding they types of analyses that can be implemented.
**Ansys Stress Analysis [1]**

The recommended Stress Analysis is listed in Ansys as the Von Mises Stress Analysis. The following procedure outlines the steps necessary to complete a stress analysis and review the results. Note: if you using composite materials in your structural analysis looking at the Maximum stresses in the X and Y directions may be more appropriate than Von Mises Stress. The following procedure describes the analysis commands in ANSYS after the desired structural model has been created. The underlined words are options in the main menu of ANSYS.

**Element Type**

- **Add/Edit/Delete**
  - Choose element type desired

**Real Constants** (A thickness will need to be specified here if you have shell elements defined.)

- **Add/Edit/Delete**
  - Choose element type desired

**Material Properties**

- Define the type of structural material you will be using.

**Mesh**

- If you are using shell elements you will need to mesh the areas only.
  - Choose a reasonable mesh.

Before conducting the analysis you must turn the Pre-Stress Calculation on. (This will be necessary for calculating buckling loads.) Type `pstres, on` in the command line.

**Apply Loads and Constraints**

**Solution**

- New Analysis
  - Static

**Solution**

- **Solve**
  - Current LS
    - (Make sure when you click this button that you are not in dynamic mode for the model window. The solution will not run!!!)

**General Postprocessing**

- **Plot Controls**
- **Contour Plot**

Executing the **Results Viewer** also allows the viewing of different types of solutions from the same drop-down window.
**Nodal Solution**

**Stress (Von Mises)**

Finish
(Type finish in the command line.)

**Ansys Buckling Analysis [1]**

A static solution (Von Mises Stress solution applies) must be conducted before a buckling solution can be executed. Pre-Stress must also be on when the static solution is solved. The finish command must be implemented after conducting another type of analysis, before the buckling analysis can be executed.

**Solution**

**Analysis Type**

New Analysis

Eigen Buckling

*Block Lanczos:* (Default. To find many modes (about 40+) of large models. Recommended when the model consists of poorly shaped solid and shell elements. This solver performs well when the model consists of shells or a combination of shells and solids. Works faster but requires about 50% more memory than subspace)

*Subspace:* To find few modes (up to about 40) of large models. Recommended when the model consists of well-shaped solid and shell elements. Works well if memory availability is limited. Requires a significantly more amount of disk space.

**Solution**

Solve

Current LS

*(Make sure when you click this button that you are not in dynamic mode for the model window. The solution will not run!!!)*

Finish
(Type finish in the command line.)

Expanding the *Eigen Buckling Solution* (So you can see find shapes.)

**Solution**

**Analysis Type**

Expansion Pass (click in the box next to Expansion Pass to turn it ‘on’)

---

4
Solution
Load Step Opts
  Expansion Pass
    Single Expand
  Expand Modes
    (Choose the number of modes to expand. Initially I would recommend 3-5.)

Solution
Load Step Opts
  Output Cntrl
    Solu Printout

Solution
Load Step Opts
  Output Cntrl
    DB/Results File

Solution
Solve
  Current LS

Finish
  (Type finish in the command line.)

Reviewing the Results
  (So you can see the mode shapes.)

General Postproc
  Summary Results
    (This provides a list of the mode numbers and eigenvalues)

General Postproc
  Read Results
    By Load Step
      (set load sub-step increment to 0.5 – this will produce results at half step increments of what was calculated.)

General Postproc
  Read Results
    Deformed Shape

To view multiple mode shapes:
  Read Results
    First Set
Second Set (Select this for each consecutive mode shape to be viewed.)

General Postproc
   Plot Results
      Contour Plot
      Nodal Solution

Interpreting the Buckling Analysis Results

4) Design Trades

Include some type of discussion on how the structural design or analysis might be improved. (Unless you've created the “perfect” structure) This information also includes any design trades conducted. You should conduct an investigation into how the resizing (length and width) of the vehicle effects how the design meets the requirements. These types of trades can help you find a ‘better’ vehicle than the initial guess. Compiling the design trades in a concise format can also help to more easily determine how the vehicle structural design (and ultimately the mass) will change as the design changes. Compiling a graph that shows length of vehicle versus maximum load; or thickness versus length (with safety factor) can be good ways to demonstrate design trades and show correct trends.

References

[1] Online Ansys Manual (Search for appropriate words corresponding to subject matter.)

[2] Schneider, Steven, P. AAE 450 Spacecraft Design; Course Structures and Loads Description, Design Methods Handout.