NASA/TM-2002-211931

A Procedure for Structural Weight Estimation of Single Stage to Orbit Launch Vehicles (Interim User's Manual)

Zoran N. Martinovic and Jeffrey A. Cerro
Langley Research Center, Hampton, Virginia

September 2002

The NASA STI Program Office ... in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA STI Help Desk at (301) 621-0134

- Phone the NASA STI Help Desk at (301) 621-0390

- Write to:
  NASA STI Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076-1320

NASA/TM-2002-211931

A Procedure for Structural Weight Estimation of Single Stage to Orbit Launch Vehicles (Interim User's Manual)

Zoran N. Martinovic and Jeffrey A. Cerro
Langley Research Center, Hampton, Virginia

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

September 2002

Available from:

NASA Center for AeroSpace Information (CASI)
7121 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 605-6000

Abstract

This is an interim user's manual for current procedures used in the Vehicle Analysis Branch at NASA Langley Research Center, Hampton, Virginia, for launch vehicle structural subsystem weight estimation based on finite element modeling and structural analysis. The process is intended to complement traditional methods of conceptual and early preliminary structural design such as the application of empirical weight estimation relationships or application of classical engineering design equations and criteria on one dimensional "line" models. Functions of two commercially available software codes are coupled together. Vehicle modeling and analysis are done using SDRC/I-DEAS , and structural sizing is preformed with the Collier Research Corp. HyperSizer program.

Table of Contents

List of Figures

List of Tables

List of Appendices

1. Introduction

This document serves as an interim user's manual for current procedures used in the Vehicle Analysis Branch at NASA Langley Research Center for launch vehicle structural subsystem weight estimation based on finite element modeling and structural analysis.

A general overview of the weight estimation procedure is presented first. It is followed by a detailed description of the procedure with recommendations on how to deal with the design process.

2. General Overview of the Weight Estimation Procedure

The process described in this report is based on application of finite element (FE) models to estimate weight of typical cylindrically shaped launch vehicles. The process is intended to complement traditional methods of structural design such as application of empirical weight estimation [1] or application of classical engineering design equations and criteria on one dimensional "line" models. Because of the requirement of fast turn-around at the early stage of vehicle design this method utilizes relatively simple three dimensional finite element models for structural weight estimation of the new and untested launch vehicle concepts.

The ultimate objective of this effort is to generate a procedure to automate structural weight estimation for new vehicle designs and to reduce the interaction required from analysts/designers to a "reasonable level" during the initial design stage. This procedure could further be integrated with other design disciplines, such as propulsion, trajectory analysis, aero and thermo analysis, into a unified code/procedure that would produce an initial launch vehicle candidate design with the low effort and in a short time.

The general outline of the procedure is shown in Figures 1 and 2. Vehicle geometry and preliminary structural weights and system weights are defined first from other sources such as The CONfiguration SIZing Program [1]. The vehicle geometry and finite element model meshing is done in I-DEAS [2]. Preliminary vehicle mass from CONSIZ is discretized and lumped to the FE model through a process which uses EXCEL spreadsheets and a JAVA program. External loads are modeled next. These are loads used to represent basic lift, thrust, control and tank pressure forces which are later combined and scaled to create vehicle design conditions. Inputs from different sources are compiled (such as from a trajectory program) and then the actual design load cases are created using a procedure based on integration of I-DEAS, EXCEL spreadsheets, text files and a JAVA program. The net result of this process is a lumped mass/mass-less shell element FE model with proper boundary conditions and static loading ready for a linear static solution.

The structural sizing part of the procedure consists of an initial sizing run which produces first estimates of vehicle stiffness and structural weight. After this, the user needs to iterate the analysis and sizing runs until desired convergence of vehicle weight is achieved. Convergence satisfies the iterative nature of calculating new strucutural elment sizes and letting this new element definition influence the vehicle mass and stiffness matrices. Static analysis is performed inside I-DEAS and results are exported to the sizing program. HyperSizer [3] sizes the vehicle shell panels to support internal running loads imported from I-DEAS. The outcome of this is a consistent mass shell vehicle ready to be imported back to I-DEAS for a new set of static analyses.

Once the iteration between I-DEAS and HyperSizer produces sufficiently converged vehicle structural weight, the process ends. Updated stiffened skin theoretical structural weights can then be modified from the theoretical state to the "as-built" weight and exported to other disciplines in the vehicle design process (such as back to CONSIZE).

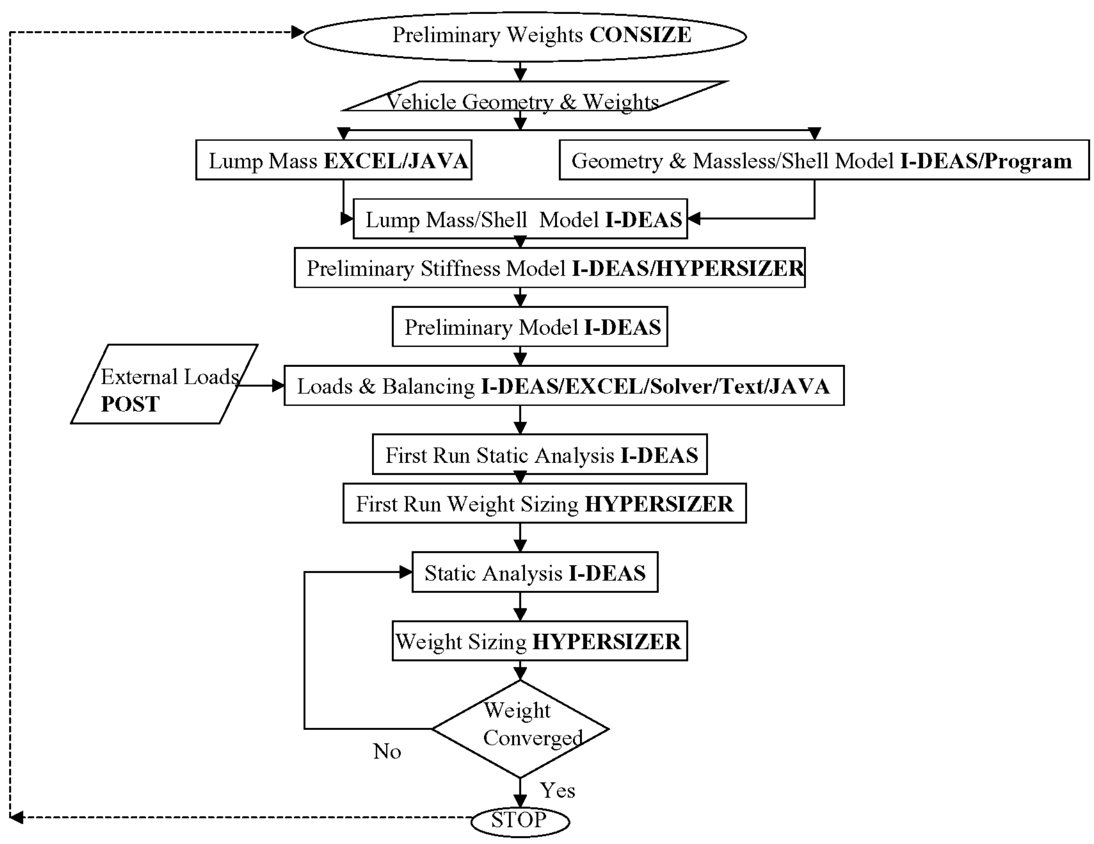


Figure 1. General outline of the procedure for structural weight estimation

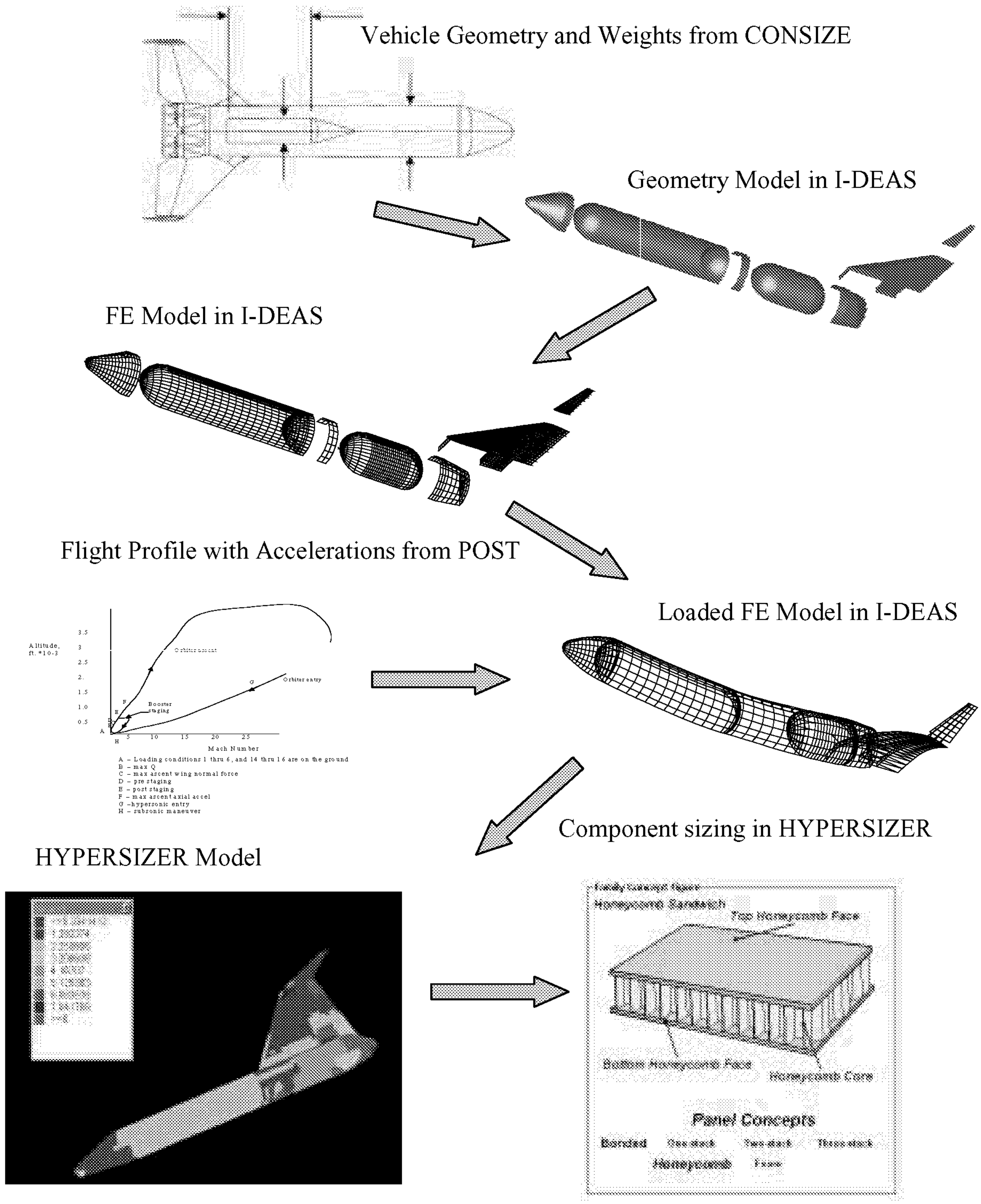


Figure 2. Graphical outline of the weight estimation procedure

3. Detailed Overview of the Weight Estimation Procedure

The whole procedure consists of the following major sub-procedures:

- Geometry and Finite Element Modeling,
- External Loads Modeling and Load Balancing,
- First Static Analysis and Structural Sizing,
- Iterative Static Analysis and Structural Sizing.

The flow chart of the detailed procedure is illustrated in Appendix A. Table 1 summarizes the software and programs used in this procedure.

Table 1. Software used to estimate vehicle weights

Software	Application
I-DEAS	Geometry and finite element modeling; static analysis.
HYPERSIZER	- Preliminary vehicle stiffness definition; - element sizing.
I-DEAS programs	Geometry and finite element meshing; physical property assignment to finite elements; lump mass distribution to finite elements. General CAD/CAE tasks.
JAVA programs	Associates finite element grids to lump masses; combines force and pressure loads. General manipulation of the ASCII file representation of the FEA entities.
EXCEL spread sheets	Summarizes system weights and associates them with finite element grids; combines “unit loads” into flight loads.
EXCEL spread sheet-solver	Flight load balancing.
Text files	Lump mass processing; force and pressure processing.

3.1. Geometry and Finite Element Modeling

The initial estimates of vehicle weights and geometry have to be acquired from other sources such as the CONSIZE program. Appendix B is a listing of output from CONSIZE that contains a breakdown of system weights into multi-level sub-system weights. This output also contains general design data and vehicle parameters with geometry information that serve as the starting point for the structural analysis.

Vehicle modeling is further divided into the following three interdependent tasks:

- Vehicle Geometry and Mass-less Finite Element Modeling Task,
- Vehicle Lump Mass Modeling Task,
- Vehicle Preliminary Stiffness Definition Task.

The final product of these three tasks is a vehicle finite element model built of mass-less shell elements and nodal lumped masses whose total weight equals the vehicle weight less the weight of main propellant. The user will notice the absence of other types of finite elements, such as beam elements, which one would expect to be present in the model beside stressed skin. The inclucsion of beam elements is a complication to the procedure as it currently stands and is being worked as a future enhancement.

3.1.1. Vehicle Geometry and Mass-less Finite Element Modeling Task

In this task vehicle geometry is generated at the structural component sub-assembly level such as: fuel tank, vehicle nose, inter-tank assembly, payload bay, thrust structure, wing, tail and winglets. Those CAD surfaces are then meshed into finite element models of the sub-assemblies. The whole modeling process can be done either in a single I-DEAS Model file or in separate Model files.

At this stage of the modeling process, finite elements do not have mass and the stiffness is defined as for a 0.001 inch thick steel element with the following material properties: modulus of elasticity of 3×10^7 lb/in^2, Poisson's ratio of 0.29 and mass density is 0.0 lb sec^2/in^4. Selection of steel and thickness was quite arbitrary.

The finite elements are then organized into groups of panels. Each shell finite element in the panel has the same physical property name assigned to it in I-DEAS. These panels are the smallest structural entities that may be later on analyzed and sized in HyperSizer. Panels represent distinct regions of a single set of manufacturing sizes. For example, a stiffened skin panel may be made up of many elements but each element will have the same stiffener arrangement and gage sizes as any other in the panel. In HyperSizer these panels are called "components". It is important that this process of associating the physical property names to the elements produces physically meaningful panels. Naming the physical properties and associating them with proper elements is therefore a very important step in the vehicle design process.

The user has three options to do vehicle geometry and mass-less finite element modeling tasks at the sub-assembly level in I-DEAS.

1. Create geometry and do meshing with the help of ready-made I-DEAS Program files.
2. Create geometry and mesh data using the I-DEAS Graphical User Interface (GUI).
3. Use available sub-assembly geometry models in so called I-DEAS "parametric form" and mesh them manually.

The first method is the simplest but least accurate one. The fuselage sub-structures are built of the simplest geometric entities such as cylinders, ellipsoids and frustum of cones. Figure 3 illustrates these sub-assemblies. Appendix C contains a typical I-DEAS Program file for a fuel tank that is used to generate geometry and finite elements for a liquid oxygen tank. The advantage of this method is that it is simple to apply. The disadvantage is that it does not cover more complicated shapes such as a droop nose.

This method can use an I-DEAS program for automated property assignment. This program is listed in Appendix D. The program runs interactively inside I-DEAS and requires the following information be provided by the user:

- Starting element number,
- Ending element number,
- Number of elements per property card,
- Property prefixes string.

The program assigns a physical property to the consecutive elements in the model. It allows further division of the sub-assembly with properties grouped around physically meaningful structural entities such as fuel tank bulkheads and barrels. It is well suited for simpler shapes such as fuel tanks, inter-tank adapters, simple nose sections and thrust structure. It should not be used for wing-like sub-assemblies and for complex shapes such as a droop nose.

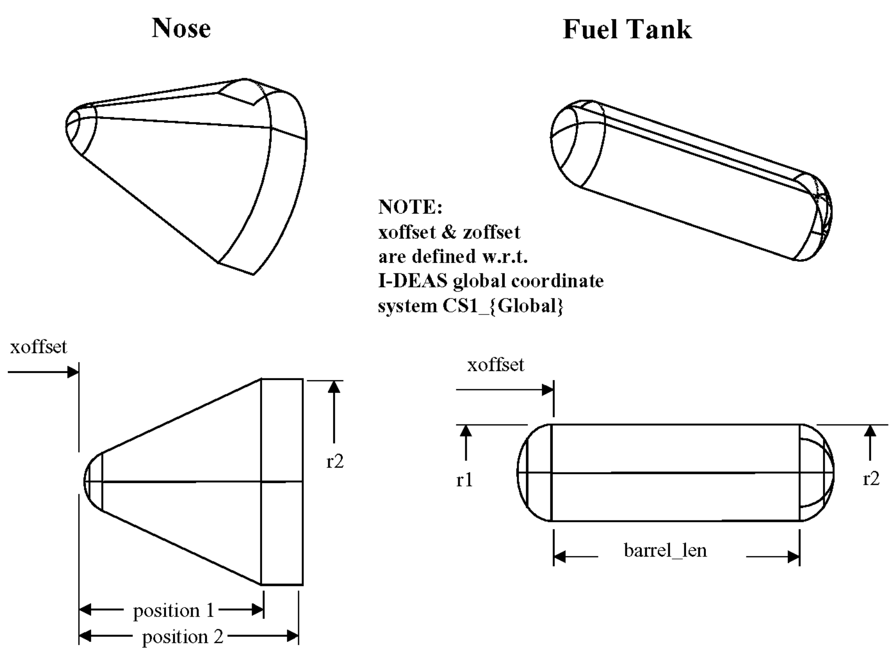


Figure 3a. Geometry of sub-assemblies generated by I-DEAS Program files

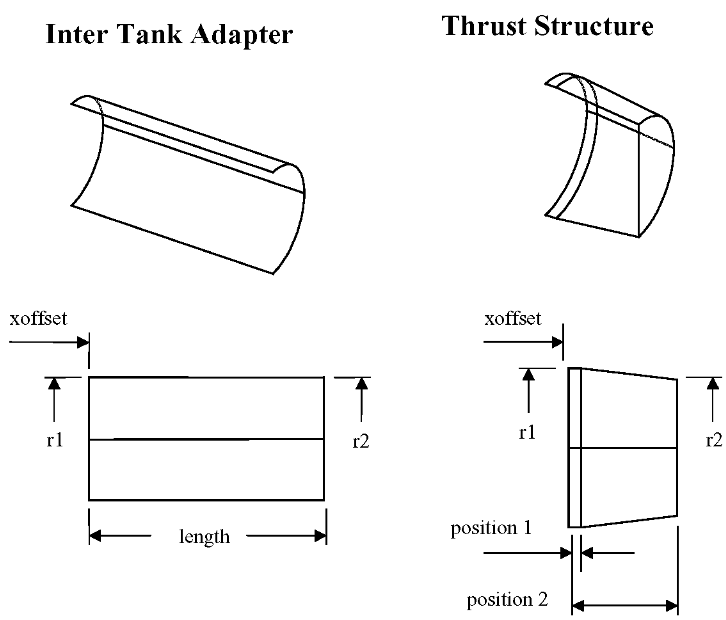


Figure 3b. Geometry of sub-assemblies generated by I-DEAS Program files

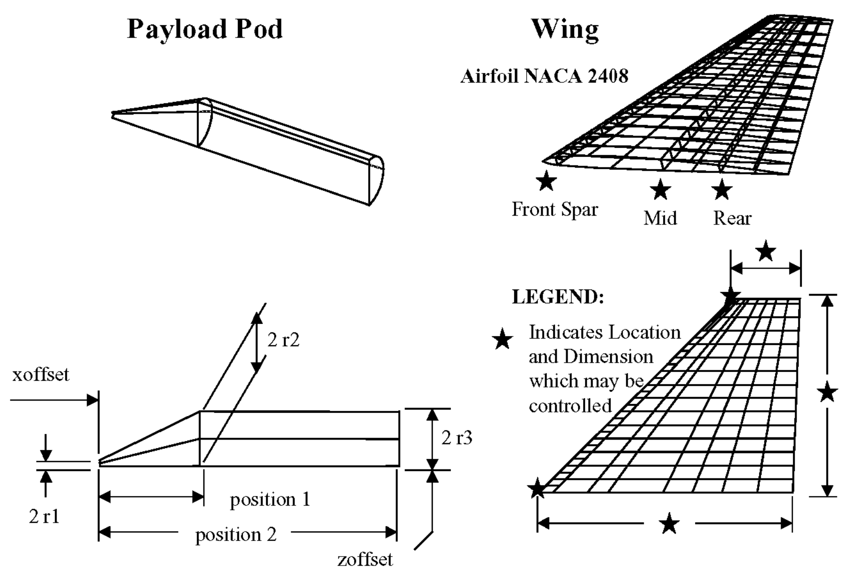


Figure 3c. Geometry of sub-assemblies generated by I-DEAS Program files

The second option allows the user full capability of the IDEAS GUI environment. Geometry, meshing and property assignment data for individual parts must be completed. The rest of the process is general enough such that user defined components can be anlayzed. A disadvantage to this method is that automation of man-in-loop process flow is not desireable.

The third method is sort of a mixture of the prior two. Prior part models of shapes more complex than have been used via the first method are stored in an IDEAS Library. These parts are retrieved with appropriate dimension values applied. A typical complex shape – droop nose section is shown in Figure 4 with variable geometric parameters shown in the figure. The library of so called I-DEAS "parametric models" could be generated ahead of time. One disadvantage of this method is that the Part Coordinate System may not be aligned with the I-DEAS Global Coordinate System and this may cause some problems in the ensuing steps if not taken care of (see Note in Figure 3a).

After all sub-assemblies have been created, they have to be assembled together into a vehicle assembly finite element model. Firstly, every sub-assembly model has to be exported from the I-DEAS Model File in Universal file format. The Universal files are then read into a new Model file one by one. Sub-assemblies generated from the parametric models should be imported last because of a problem with their coordinate system. This process will create new parts inside the Model file associated with each sub-assembly. Each part has also a finite element model associated with it. The FE models are separated and need to be assembled and appended into a vehicle assembly model. This process generates a few identical nodes at the interfaces between the parts. These nodes must be "equivalenced" (i.e. merged together) to provide structural continuity between the parts. Note that this assembly process was created in IDEAS V6. New code features, such as assembly FEA modeling in I-DEAS, may be taken advantage of as long as the intent of the process presented here is preserved.

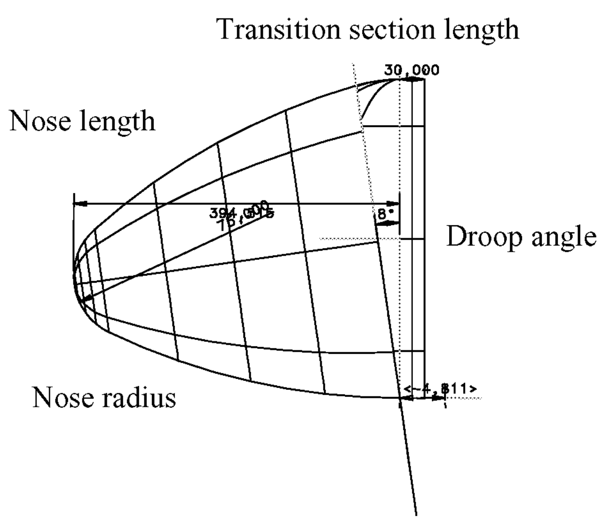


Figure 4. Droop nose geometry

Connection between the wing and fuselage is modeled with Rigid Links or mass-less FE Beams with realistic stiffness properties. These elements will not be sized in HyperSizer and are in the model only to transfer load from the wing or tail surfaces into the fuselage. The detail of such a connection is shown in Figure 5. Care should be taken such that the rigid links tend to simulate phyisical connections that the joined parts would see in an actual assembly.

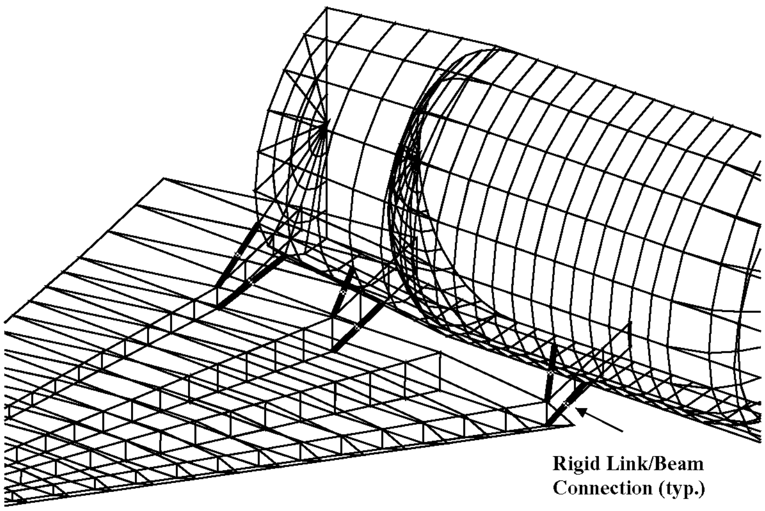


Figure 5. Rigid link or mass-less beam connection between wing and the fuselage

Finally, the model has to be prepared for preliminary vehicle weight definition and distribution. Densities of all materials in the model must be zeroed. Finite element nodes can be grouped into spatial groups which should correspond to the different vehicle systems listed in the CONSIZE output of Appendix B. The spatial grouping should be done so that the center of gravity of the group is as close as possible to the location of the center of gravity of particular system defined in CONSIZE output. The group names should be different from element property names.

The final product of this task is a vehicle finite element model with no mass and with arbitrary stiffness. Appearance of such a model is shown in Figure 6. This model has to be exported in I-DEAS Universal file format and will be used as an input file during the process of preliminary vehicle mass definition.

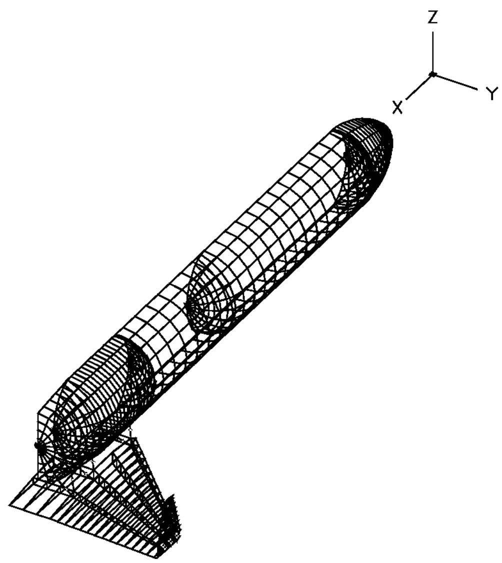


Figure 6. Finite element model of a launch vehicle

3.1.2. Vehicle Lump Mass Modeling Task

System weights from CONSIZE output (see Appendix B) have to be mapped to appropriate finite element grid points. This can be done in two steps. Firstly, weights are parsed either to the groups of grid points or to the region of grid points in the EXCEL spreadsheet. Text file format of such a spreadsheet is shown in Appendix E and the abbreviated form of that file is shown in Table 2. This file will be used as input into a JAVA program that maps the weights from CONSIZ to I-DEAS nodal masses.

Table 2. Abbreviated form of a CONSIZE_MOD Text file

CONSIZ Component	FEA Group	Weight (lbs)	Mapping	X Begin (inch)	X END (inch)
begin_components					
vert_fin	vertical_tail	4041	component	0	0
payload_bay	fuselage_side	6086	fofx	1574	1974
end_components					

This spreadsheet/text file must begin with the statement "begin_components" in the first column and it must end with the statement "end_components" in the same column. Between these two statements, the user may establish the relationships between weights copied from the CONSIZE output and the finite element nodes. The first column also contains vehicle system CONSIZE names for a reference purpose only. Finite element model groups must be entered in the second column precisely the same way they were named in the model. There is one naming rule for the CONSIZE Component and the FEA Group:

- the CONSIZE Component should not be named the same as any finite element Physical Property Set data.

The "Weight" column contains the weights in pounds from CONSIZE. The user has two options to map the CONSIZE weight onto model nodes. The "component" entry in the "Mapping" column will allow the JAVA program to map, for example, 4041 lbs of vertical fin weight to all nodes contained in the FE group "vertical_tail". Thus, the weight will be spread in form of lumped masses to all nodes of the particular group. The "fofx" entry will instruct the program to spread the weight only on the sub-set group of nodes which starts at location X-Begin inches in the I-DEAS Global Coordinates and ends at X-End. For example, the payload bay weight of 6086 lbs will be mapped between Station 1574 and Station 1974 on all nodes belonging to a group named the "fuselage_side". Note that the nodal mapping is currently slightly inaccurate as mass will tend to concentrate where nodal density concentrates. Future versions of the process intend to use an areal spreading of the component masses and calculate nodal masses based on such a distribution.

Next, the user has to run the JAVA program "consiz2unv" to distribute vehicle weights according to the mapping plan set in the spreadsheet. This program requires two files as input: a a text version of the EXCEL mass mapping data file, and a Universal file created from the current I-DEAS Model file. The output from this program is an I-DEAS Program file with vehicle masses lumped at model nodes. The last step in this task is to run the Program file inside I-DEAS to add the lumped masses to the model. Appendix F lists the JAVA code command to run the program. Parameter $1 of this input command is the name of the universal file, $2 is the mass mapping data file, and $3 is the name of a mass assignment program file that will be created.

The final product of this task is vehicle finite element model with all dry vehicle mass lumped at the nodes. The weight of the main fuel will be modeled as a time dependent pressure loading condition. A good check that all mass has been assigned to the I-DEAS finite element model is to check the model inertia properties in the I-DEAS Model file.

3.1.3. Preliminary Vehicle Stiffness Definition Task

At the end of this task all shell finite elements will have default stiffness properties assigned to them. The tasks consists of the following three steps.

Static analysis of the vehicle model exposed to an arbitrary load and restraint condition has to be done first in I-DEAS. This analysis may be "arbitrary" because HyperSizer will first be run in an analysis (not sizing) mode to setup initial element stiffnesses. The goal of this analysis is only to create an I-DEAS model in Universal file format which is readable by HyperSizer. Application of loads and boundary conditions is irrelevant in the sense that the loading condition is unimportant to HyperSizer at this point but it is necessary to keep the Universal file in a format HyperSizer can deal with. The finite element model and results of this analysis should be in "Inch (pound f)" units. Before running the static solution analysis "Element Force" and "Shell Stress Resultant" should be selected as output results in I-DEAS. A Universal file with the model and the results is exported after the analysis.

Stiffness definition takes place in HyperSizer. The user is encouraged to read the HyperSizer Manual for detailed instructions on how to run the program in conjunction with I-DEAS finite element analysis. The procedure flow chart in Appendix A may be used as a guide for this particular process. The Universal file from the previous step has to be imported into the HyperSizer database. A vehicle material and a sandwich panel as a structural family need to be selected next. Note that HyperSizer offers a large selection of structural panel design concepts (i.e. families). For simplicity the current procedure uses only sandwich panels. This is an obvious limit of the procedure and can be overridden as the user gains expertise with HyperSizer and I-DEAS. All HyperSizer Components (where a Component is a group of shell finite elements with same physical properties in I-DEAS) need to be grouped into a single HyperSizer Group. The sandwich panel thickness of the Group needs to be defined. A default thickness of one inch is recommended (0.1 inch for the face sheets and 0.8 inch for the core). The Group variable (i.e. thickness) range should be set to a single value and permutation set to one. This is because there will not be a sizing run at this stage in the procedure. After setup of the HyperSizer model is complete and the program analysis option has been executed the properties and materials (i.e. stiffness and weights) of Components are exported by HyperSizer in I-DEAS Universal file format. Note that the I-DEAS Universal file output is generated only when the entire Project is analyzed.

Before reading the Universal file into a new I-DEAS Model file the consistent mass matrix of the shell elements has to be edited out. The updated vehicle FE model has preliminary stiffness defined for all mass-less shell elements, and all preliminary structural weights and system weights (with the exception of the main fuel weight) are modeled as lumped masses.

3.2. External Load Modeling and Load Balancing

The user must define a set of design load conditions that the vehicle model will be subject to. These load conditions may be such as: vehicle on the pad, liftoff, maximum dynamic pressure in flight, maximum thrust, main engine cut-off, re-entry and so on. Table 3 lists a set of load cases typically used for weight estimation. Information about these loads may be available to the user from different sources and programs.

Table 3. Design conditions

1) Proof
2) 10 day wind on pad
3) 1 day wind on pad
4) liftoff
5) Max Q
6) Max Fn
7) Max Axial acceleration
8) Subsonic Entry Manever
9) Main-Gear Touchdown
10) All –Gear Touchdown
11) Ground Handling

The procedure to create design loads is based on so called "Unit Load Sets" which are the simplest load building blocks of the design loads. Unit load sets are scaled, combined and balanced to create the actual design case loads listed in Table 3. The design loads are then used in the static analysis for element sizing. The process of creating design loads consists of several tasks (see Appendix A):

- Generation of the Unit Load Sets,
- Combination of the Unit Loads into flight loads and load balancing,
- Processing of force and pressure loads,
- Final assembly of loads into load conditions.

3.2.1. Unit Load Set Task

The most basic load sets are built in this task in I-DEAS. Their formulation and modeling is left to the user's decision as long as the user is applying them consistently though the rest of the procedure. They may be defined as a unit pressure of 1 (psi) per element, as illustrated in Figure 7 for the "unit" wing lift, or as a unit force of 1 pound distributed to all thrust structure end bulkhead rim nodes. The Unit inertial loads are defined as properly oriented 1 G linear acceleration. The Unit rotational acceleration may also be defined. These load sets will be used whenever flight conditions require modeling of loads with variable magnitude.

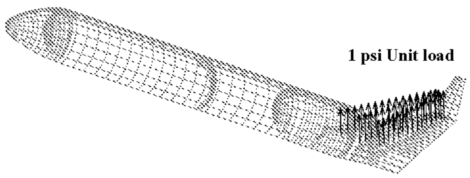


Figure 7. Unit Load for wing lift

Some loads may be modeled either as a unit load or they may be modeled with actual magnitude. Wind pressure on the vehicle on a launch pad is a load that often is modeled as a precalculated input surface pressure.

The fuel head pressure on the tank walls can not be modeled as a scaleable unit load. It must be modeled separately for each flight case That is because it is a time dependent load due to the continual use of fuel and changing acceleration vector throughout the ascent trajectory. Figure 8 shows resulting pressure vectors on a tank wall as calculated for a typical fluid acceleration condition. Appendix G lists an I-DEAS Program file for automatically creating tank head pressure loads. Table 4 shows the input part of the same file with input lines that have to be edited by the user. Note 5 in Table 4 indicates that all tank finite elements must be grouped in the I-DEAS Model file in appropriate tank groups.

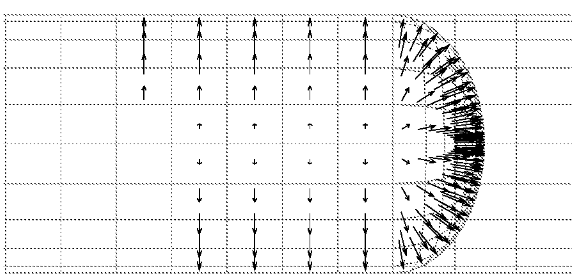


Figure 8. Tank head pressure

Table 4. Input lines from a tank head pressure modeling I-DEAS Program file

```
C : Start USER INPUT _________________________
C : position positive point on neg side to reverse pressure sign
K : /options global_symbols on
K : enter ldst_name
K : yes
K : "lh2_40%_2.93_0_0.17"     This is a name of the load set as
                               it will appear inside I-DEAS.
K : /options global_symbols on
K : enter grp_name
K : yes
K : "lh2_tank_elements"  This is a shell elements group name
                         which must be identical to the one
                         previously defined in I-DEAS.
                         See Note 5) below.
K : # ullage=.0
K : # rho_g=.0075042     See Note 3) below.
C : Vehicle acceleration
K : # ax=-2.93           See Note 2) below.
K : # ay=.0
K : # az=-0.17           See Note 2) below.
K : # declare pos(3)
K : # pos (1)=1441.      See Note 1) below.
K : # pos(2)=.0
K : # pos(3)=.0
C : END USER INPUT _________________________
```

Following are the instructions how to organize input in that file.

1. Establish amount of fuel that remains in the tank and position of the fuel surface along the vehicle centerline in the Model Global Coordinate System.
2. Determine components of axial acceleration (i.e. along vehicle axis) and normal acceleration (i.e. Z axis of Figure 6). Express these components in unit gravitational acceleration (Gs).
3. Calculate magnitude of the resultant acceleration in Gs and multiply that number with fuel density in lb/in3.
4. Run the Program file.
5. Two head pressure regions will be generated. The first one is a “wetted” region which covers correct fuel tank elements bellow the fuel surface line (see Figure 8). The second region of elements is the erroneous one and it has to be graphically edited out in the I-DEAS Model file. This region is easily identifiable because the pressure arrows are directed in the opposite direction from those shown in the “wetted” region.

These Unit Load Sets are applied one at the time to the free-free FE model of the vehicle and static analyses are performed. I-DEAS Listing text files (.lis) and a Universal file (.unv) from these runs are saved for the next steps. The Listing files contain sum of applied loads and moments along the reference (Global) axes and the origin respectively (see Table 5).

Table 5. Unit Load run results in an I-DEAS Listing file

```
NET APPLIED LOAD FOR LOAD SET 6 - LH2 1PSI INTERNAL PRESURE
   FX = -3.61899D-11, FY =  4.60200D+05, FZ = -2.24052D-10
   MX = -3.17788D-09, MY = -3.42105D-08, MZ =  4.22512D+08
   MOMENTS TAKEN ABOUT THE ORIGIN
```

3.2.2 Combination of Unit Load Sets into Flight Loads and Load Balancing

This is the first stage during which the unit loads are combined and actual vehicle design load conditions are generated. All steps are done using an EXCEL spreadsheet. One spreadsheet per each load condition must be set. A typical load balancing spreadsheet is shown in Figure 9.

Resulting three forces and three moments from the Unit Loads analysis are copied to the respective Unit Load entry in the spreadsheet and scaled to physically meaningful magnitudes for a particular load condition (see Notes 2 and 3 in Figure 9). There is a list of more than twenty vehicle load sub-sets such as: lift, thrust, aero, nonstructural inertia loads, structural inertia loads, fuel loads etc. Some of the scaling factors can be predefined and are based on known load magnitudes during the vehicle flight stage or during vehicle ground operations on the launch pad. Input, such as vehicle acceleration, from other programs such as POST – a trajectory optimization program [4] may be used (see Appendix A). For unconstrained type flight conditions the other scaling factors must be calculated during the vehicle balancing process so that, applied normal forces, axial forces and pitching moments sum to zero (see Notes 4 and 6 in Figure 9). Therefore, these scaling factors, usually two to three, are treated as variables. This requires application of the EXCEL spreadsheet Solver (see Figure 9, Note 5). Solver is an optimization program which varies selected load scaling factors to achieve zero pitch moment subject to zero constraints on net axial and net normal force and subject to other constraints on flight loads. Outputs from the Solver are all computed scale factors. Constrained load conditions, such as having the vehicle exposed to wind on the launch pad, do not need to be balanced. All flight load conditions must be balanced.

Some flight load conditions, such as lift-off with impact considerations due to rapid thrust build-up and hold-down release may require special treatment. A simplistic calculation of a dynamic thrust factor for the liftoff condition is shown in Appendix H. Users may desire to calculate this effect with other methodologies. The liftoff acceleration must be multiplied by the dynamic magnification factor before being applied in the spreadsheet.

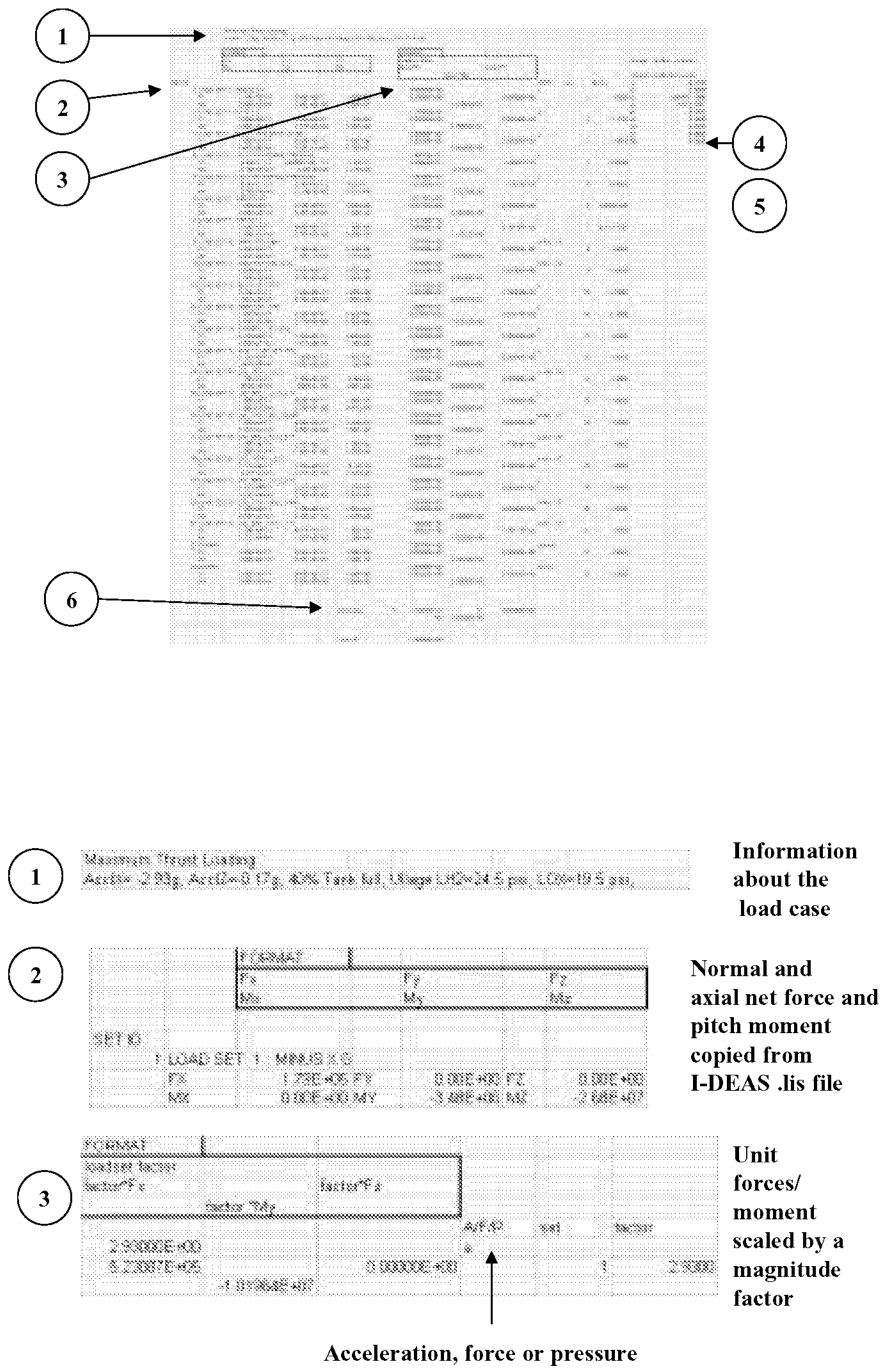


Figure 9a. Typical load balancing spreadsheet

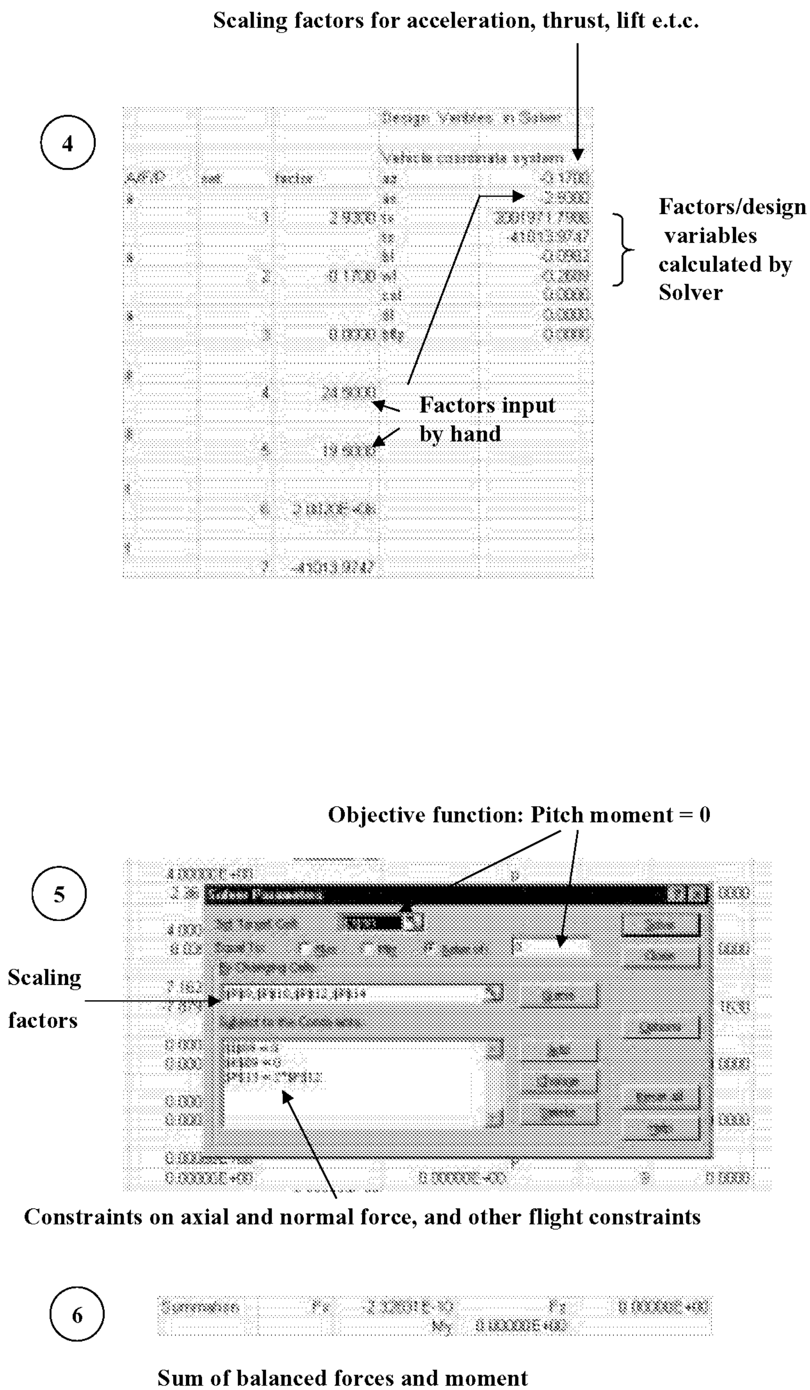


Figure 9b. Typical load balancing spreadsheet

3.2.3. Further Processing of Force and Pressure Loads

The goal of this task is to organize design load case data into a format readable by I-DEAS. The load scaling factors that were obtained in the EXCEL spreadsheets are applied to three distinct groups of loads. These loads are then combined together into I-DEAS load conditions. The three groups of loads are:

- Force loads
- Pressure loads
- Acceleration loads

Multiple force or pressure loads are combined into a single I-DEAS loadset with a utility computer program. The unit force (or pressure) loads are scaled and combined into force (or pressure) model flight loads by a JAVA program (see Appendix F for the program execution path). The program called "combine_loads" requires two input files: the I-DEAS Universal file which contains the Unit load definitions, and a text file which brings in information about scaling factors and defines which unit loads need to be processed. Listing of this text input file is shown in Table. 6.

Table 6. An input file for the JAVA "combine_loads" program

```
zoran-wb004c-2.unv Name of I-DEAS .unv file with Unit loads
pressure This file will be use to combine pressure (force)
4     Total number of Unit loads to combine
104   A new load set numher defined for I-DEAS
liftoff combined pressure Name of a new load set in I-DEAS
6 10.3 7 22.0 17 1.36 21 1.36
                                   Number and a scaling factor of
                                   a Unit load
```

The output from JAVA "combine_loads" is a new I-DEAS Universal file that consists of a single combined force (or pressure) loadset.

The Unit inertia (i.e. acceleration) loads have to be scaled in I-DEAS by the scale factors obtained in the EXCEL spreadsheets as appropriate for each final design load condition

3.2.4. Final Assembly of Loads into Load Conditions

The three scaled load sets, i.e. the force, the pressure and the acceleration are then combined into the unified load condition in I-DEAS. This load condition is used in a Boundary Condition set and subsequently for a static solution set within I-DEAS so that the static analysis may be performed.

3.3. The First Static Analysis and Structural Sizing

The initial FE model of the launch vehicle will be subject to a number of static load conditions and preliminary internal running load distributions will be obtained at this stage. The internal loads will be used to perform the first structural sizing of the vehicle. A model of the newly sized vehicle, which has new stiffness and structural mass distribution, will be ready for a series of analysis/sizing iterations whose goal is to produce minimum structural weight design of the vehicle. This section deals only with the first in the series of analysis/sizing steps.

The process employs two commercially available softwares whose functions are coupled together. The static analysis is done as a natural continuation to modeling in the I-DEAS Model file. The structural element sizing is preformed in HyperSizer. The user is advised to become familiar with these two programs. Only general outline of program capabilities and specifics related to this procedure will be covered in this report.

3.3.1. The First Static Analysis

Analysis is done in I-DEAS. After the final load sets are generated, three more preparation steps are required. The Restraint Sets are built which define general boundary conditions of the vehicle such as: vehicle on the launch pad, landing gear and free-free boundary condition. The FE model does not have beam elements to model frames for concentrated load application to the model. Therefore, the user must carefuly apply restraints, such as a nose gear restraint shown in Figure 10, to avoid application of concentrated loads normal to the plane of shell elements. Since only half of the model has been generated, symmetric boundary conditions must be modeled. Then, the Load Sets and Restraint sets are combined in the Boundary Conditions. Finally, a static analysis is selected in the Solution Sets.

There are two requirements imposed by HyperSizer on the IDEAS Universal file format:

- "Element Force" and "Shell Stress Resultant" outputs must be selected before running the FE analysis,
- the units must be changed to "Inch (pound f)".

The model and the results should be exported into an I-DEAS Simulation Universal file after the static analysis.

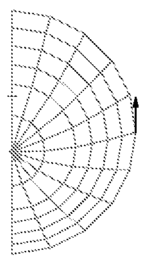


Figure 10. Nose gear restraint

3.3.2. The First Structural Sizing

HyperSizer integrates in a single tool structural design and analysis processes that are required to size a structure. Finite elements are grouped together into the smallest practically manufacturable pieces of structure called HyperSizer structural Components. Generally, Components may be either panel or beam concepts which can be analyzed and optimized subject to the imported running loads from I-DEAS and the pre-defined boundary conditions. Analyses include traditional industry methods and modern analytical and computational solutions. The optimization includes material selections and all of the dimensional variables such as panel and beam shapes, thicknesses, stiffening webs and flanges, spacings, and depths. The Components are organized into Groups that have the same initial input design parameters. Each Group belongs to a structural Family. Structural Families include broad definitions for panels and beams such as the “Unstiffened plate/sandwich family”, the “Corrugated stiffened family” etc. Within each Family there are several choices available which finalize the construction details of a concept.

The present launch vehicle structural design has utilized only the sandwich family with face sheets of equal thickness and of isotropic material. The current procedure does not support the use of other families or of beams. This is because there has not been an attempt to define material orientation vectors for stiffened skin panels in the I-DEAS-based part of the procedure, and because the beam finite elements are not generated in an automatic way and therefore are not available. All of the Groups are organized into a Project that contains all information about the structure including the finite element mesh and loads.

Following are the general steps for coupling FE analysis with HyperSizer.

- **Project Preparation** (create a Project, select the materials, setup project form).
- **FE Analysis Import** (import FE model, check and combine load sets).
- **Pre-Sizing Preparation** (select structural Family, assign FEM structural Components to Groups, select sizing variables and materials, build Assemblies, define panel buckling geometry).

After HyperSyzer analyses the Components of a Group, each will have unique design variables based as required upon Component loading and failure mode analysis. Groups can be reorganized at any time during the analysis if the initial grouping requres revision. Once the Groups have been established, each Component must be properly defined. This means the design variable ranges, material choice options, and failure mode options for each group must be input. Related to a panel based failure mode are panel length and width. These values will be read in directly from the input file, but it may be necessary to adjust the nubers to represent the stiffener direction or change in a span dimension assumed by the program. The honeycomb structural concept is used here for weight estimation purposes. Honeycomb is easy to work with in terms of structural analysis setup in HyperSizer and was sufficient to define data flow requirments. Future growth of this analysis procedure will incorporate the full sizing capability of HyperSizer.

- **Preliminary Sizing of a Component** (single analysis on a Component, import FEA running loads and review them).
- **Final Preparations** (select failure criteria and boundary conditions, select limits on variables, check loads, pressure and FEA computed moments).

Three Group design variable ranges must be defined, top face thickness, bottom face thickness, and core thickness. The user is advised to define the minimum gauge both for the face sheets and for the core. Selection of the minimum and maximum group variable bounds and the number of permutations may be guided by some industry standards. Proper selection of the variable bounds will ensure a minimum weight solution. Note that these bounds apply to the whole Group, which is an important consideration to have in mind when whole Project needs to be sized.

The user may adjust safety factors and failure modes. There are several factors in HyperSizer that may be set. By going into the “Design-to-Loads” tab the user can set a required margin of safety (MOS), a mechanical design limit load factor, and a mechanical design ultimate load factor. These setings are all very important when trying to simulate durability requirements.

- **Size and Iterate** (size Assembly/Group, check safety margins and limits on variables).

The recomended procedure is to size an Assembly first and check which Group has the Margins of Safety violated. This is an indication of an undersized structure. If a Group has minimum group variable bound reached, that is an indication of possible oversized structure. This requires that the variable bounds/permutations be adjusted.

- **Size the Project** .

After all of the setup of the HyperSizer model is completed and all Assemblies/Groups are sized, the properties and materials (i.e. stiffness and weights) of the optimized structural components are exported by HyperSizer in I-DEAS Universal file format. The I-DEAS Universal file output is generated only when the entire Project is sized. The user should check that the proper field [i.e. “FEM Properties and Materials Filename (created by HyperSizer)”] in the "Project Setup" form is filled up before sizing the Project.

3.4. Iteration between Static Analysis and Structural Sizing

The results of the first analysis and sizing are based on too many initial assumptions to be final. It is necessary to repeat the whole process to a satisfactory vehicle weight convergence and to a final vehicle design.

The iteration procedure is very similar to the first analysis and sizing process but there are also a few differences (see Appendix A).

A new I-DEAS Model file should be generated and the I-DEAS Universal file which was produced in the previous iteration need to be imported.

In the first iteration cycle only, the structural lumped masses which duplicate newly obtained shell/panel weight must be deleted from the I-DEAS Model file as obsolete.

If the variable bounds and permutations in HyperSizer were well posed during the first analysis and sizing cycle, then there will be only small changes of the bounds in the successive iterations.

4. Integration With Other Vehicle Analysis Tools

Upon successful structural weight estimation, the vehicle weight results may be passed back to the original codes (such as CONSIZE), from which they initiated, for update of input to these codes. Output from these codes, in form of vehicle geometry and new weight distribution and new external loads, will be a new input into the Structural Weight Estimation Procedure. This proces may continue until there is satisfactory convergence of the vehicle design.

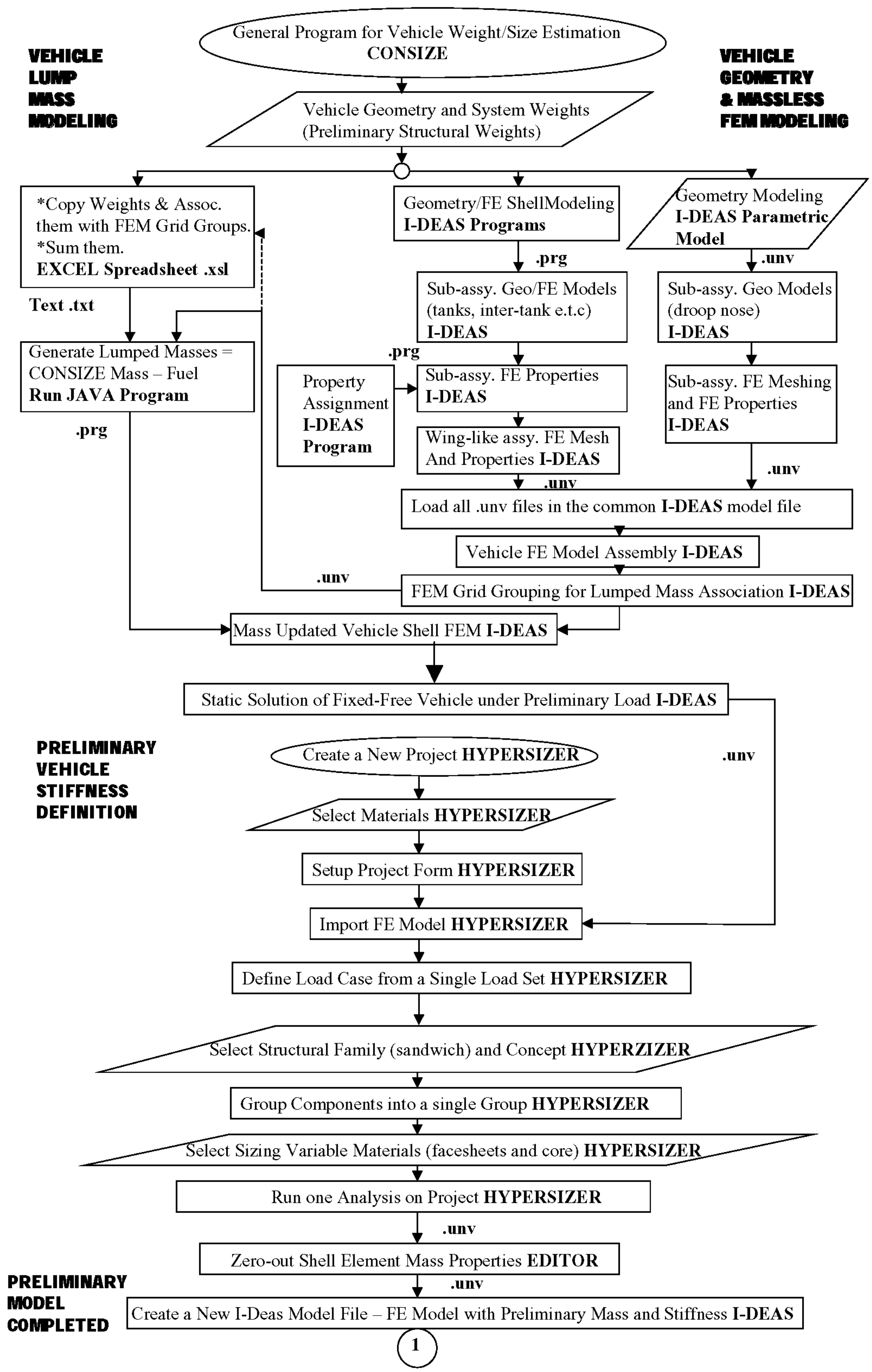
5. Conclusion

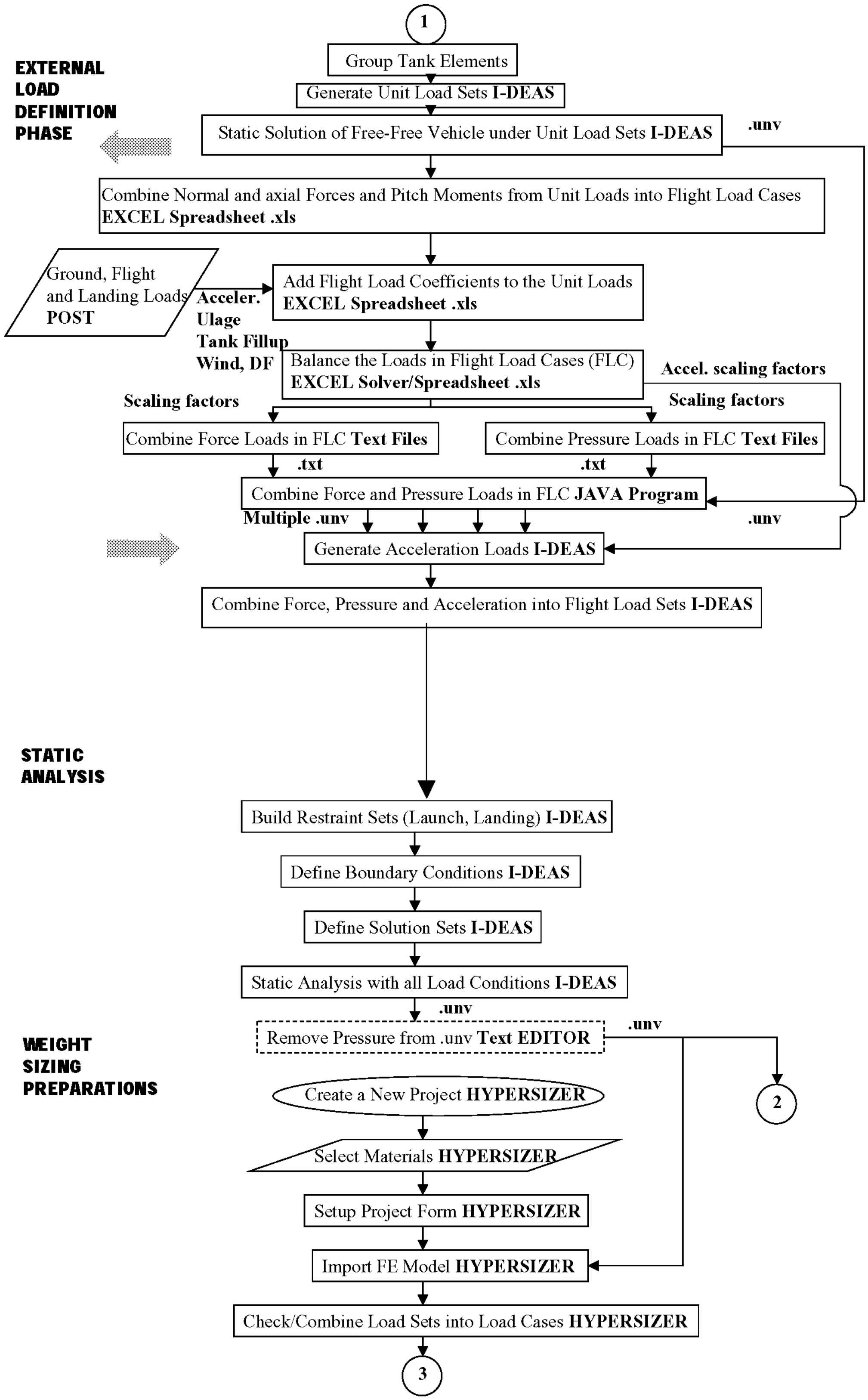
A process guide for utilization of 3D FEA and a commercial structural component design program in launch vehicle structural weight estimation has been presented. The guide is being presented at this time so users and developers of the technique have documented knowledge of the steps involved. There are indicated places in the procedure which currently are not highly rigorous, especially with regard to mass distribution, element property region assignment, and airload distributions. This was done at the expense of having a more timely document. The document defines the analysis stages, data flow requirements, and presents them to those who will help implement the system in a more automated fashion. Current users of the process can be more exacting in specific areas at their own discretion, the general procedure should still be applicably. The process starts with a vehicle configuration and weight statement. It ends with a structural weight estimation based upon static strength analysis of a shell element representation of the vehicle. Such analysis capability provides weight sensitivity to structural arrangement, structural concept, material property, and design load variations. Because of complexity, the procedure is prone to user made mistakes. An effort to automate this process is underway and should reduce both the number of mistakes and the analysis time. Ehancements in the way of having more detailed finite element modeling and external load definition are also planned for the future.

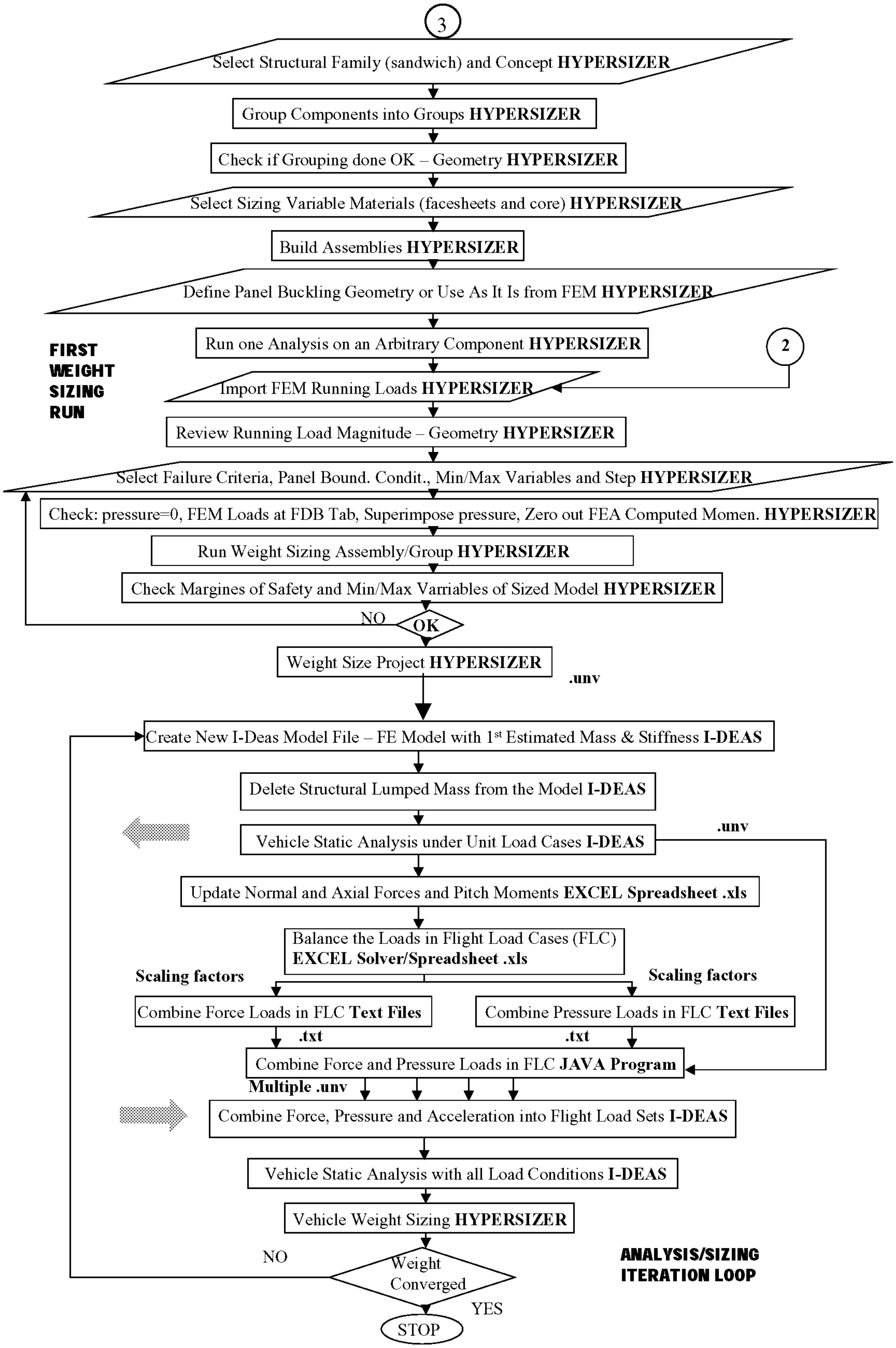
6. References

1. Lepsch, R. A., Jr., Stanley, D. O., Cruz, C. I., and Morris, S.J., Jr., “Utilizing Air-Turborocket and Rocket Propulsion for a Single-Stage-to-Orbit Vehicle,” Journal of Spacecraft and Rockets, Vol. 28, No.5, 1991, pp. 560-566.
2. Mark H. Lawry, “I-DEAS Master Series, Mechanical CAE/CAD/CAM Software”, Student Guide, Structural Dynamics Research Corporation, Milford, Ohio, 1994.
3. “HyperSizer, Structural Sizing Software”, User’s Manual, Version 1, Collier Research & Development Corporation, Hampton, Virginia, 1996.
4. R.W. Powell, S.A. Striepe, P.N. Desai, and R.D. Braun, NASA Langley Research Center, Hampton, Virginia, and G.L. Brauer, D.E. Cornick, D.W. Olson, F.M. Petersen, and R. Stevenson, Martin Marietta Corporation, Denver, Colorado, “Program To Optimize Simulated Trajectories (POST)”, Utilization Manual, Volume II, Version 5.2, October 1997.

Appendix A. Detail Outline of the Procedure for Structural Weight Estimation


VEHICLE LUMP MASS MODELING
General Program for Vehicle Weight/Size Estimation CONSIZE
VEHICLE GEOMETRY & MASSLESS FEM MODELING
Vehicle Geometry and System Weights (Preliminary Structural Weights)
*Copy Weights & Assoc. them with FEM Grid Groups. *Sum them. EXCEL Spreadsheet .xsl
Geometry/FE ShellModeling I-DEAS Programs
Geometry Modeling I-DEAS Parametric Model
.prg
.unv
Text .txt
Sub-assy. Geo/FE Models (tanks, inter-tank e.t.c) I-DEAS
Sub-assy. Geo Models (droop nose) I-DEAS
Generate Lumped Masses = CONSIZE Mass – Fuel Run JAVA Program
.prg
Property Assignment I-DEAS Program
Sub-assy. FE Properties I-DEAS
Sub-assy. FE Meshing and FE Properties I-DEAS
.prg
Wing-like assy. FE Mesh And Properties I-DEAS
.unv
.unv
Load all .unv files in the common I-DEAS model file
Vehicle FE Model Assembly I-DEAS
.unv
FEM Grid Grouping for Lumped Mass Association I-DEAS
Mass Updated Vehicle Shell FEM I-DEAS
Static Solution of Fixed-Free Vehicle under Preliminary Load I-DEAS
PRELIMINARY VEHICLE STIFFNESS DEFINITION
Create a New Project HYPERSIZER
.unv
Select Materials HYPERSIZER
Setup Project Form HYPERSIZER
Import FE Model HYPERSIZER
Define Load Case from a Single Load Set HYPERSIZER
Select Structural Family (sandwich) and Concept HYPERZIZER
Group Components into a single Group HYPERSIZER
Select Sizing Variable Materials (facesheets and core) HYPERSIZER
Run one Analysis on Project HYPERSIZER
.unv
Zero-out Shell Element Mass Properties EDITOR
.unv
PRELIMINARY MODEL COMPLETED
Create a New I-Deas Model File – FE Model with Preliminary Mass and Stiffness I-DEAS
1


EXTERNAL LOAD DEFINITION PHASE
1
Group Tank Elements
Generate Unit Load Sets I-DEAS
Static Solution of Free-Free Vehicle under Unit Load Sets I-DEAS
.unv
Combine Normal and axial Forces and Pitch Moments from Unit Loads into Flight Load Cases EXCEL Spreadsheet .xls
Ground, Flight and Landing Loads POST
Acceler. Ulage Tank Fillup Wind, DF
Add Flight Load Coefficients to the Unit Loads EXCEL Spreadsheet .xls
Balance the Loads in Flight Load Cases (FLC) EXCEL Solver/Spreadsheet .xls
Accel. scaling factors
Scaling factors
Scaling factors
Combine Force Loads in FLC Text Files
Combine Pressure Loads in FLC Text Files
.txt
.txt
Combine Force and Pressure Loads in FLC JAVA Program
Multiple .unv
.unv
Generate Acceleration Loads I-DEAS
Combine Force, Pressure and Acceleration into Flight Load Sets I-DEAS
STATIC ANALYSIS
Build Restraint Sets (Launch, Landing) I-DEAS
Define Boundary Conditions I-DEAS
Define Solution Sets I-DEAS
Static Analysis with all Load Conditions I-DEAS
.unv
WEIGHT SIZING PREPARATIONS
Remove Pressure from .unv Text EDITOR
.unv
2
Create a New Project HYPERSIZER
Select Materials HYPERSIZER
Setup Project Form HYPERSIZER
Import FE Model HYPERSIZER
Check/Combine Load Sets into Load Cases HYPERSIZER
3



3
Select Structural Family (sandwich) and Concept HYPERSIZER
Group Components into Groups HYPERSIZER
Check if Grouping done OK – Geometry HYPERSIZER
Select Sizing Variable Materials (facesheets and core) HYPERSIZER
Build Assemblies HYPERSIZER
Define Panel Buckling Geometry or Use As It Is from FEM HYPERSIZER
FIRST WEIGHT SIZING RUN
Run one Analysis on an Arbitrary Component HYPERSIZER
2
Import FEM Running Loads HYPERSIZER
Review Running Load Magnitude – Geometry HYPERSIZER
Select Failure Criteria, Panel Bound. Condit., Min/Max Variables and Step HYPERSIZER
Check: pressure=0, FEM Loads at FDB Tab, Superimpose pressure, Zero out FEA Computed Momen. HYPERSIZER
Run Weight Sizing Assembly/Group HYPERSIZER
Check Margines of Safety and Min/Max Varriables of Sized Model HYPERSIZER
NO
OK
Weight Size Project HYPERSIZER
.unv
Create New I-Deas Model File – FE Model with 1st Estimated Mass & Stiffness I-DEAS
Delete Structural Lumped Mass from the Model I-DEAS
Vehicle Static Analysis under Unit Load Cases I-DEAS
.unv
Update Normal and Axial Forces and Pitch Moments EXCEL Spreadsheet .xls
Balance the Loads in Flight Load Cases (FLC)
EXCEL Solver/Spreadsheet .xls
Scaling factors
Scaling factors
Combine Force Loads in FLC Text Files
Combine Pressure Loads in FLC Text Files
.txt
.txt
Combine Force and Pressure Loads in FLC JAVA Program
Multiple .unv
Combine Force, Pressure and Acceleration into Flight Load Sets I-DEAS
Vehicle Static Analysis with all Load Conditions I-DEAS
Vehicle Weight Sizing HYPERSIZER
NO
Weight Converged
ANALYSIS/SIZING ITERATION LOOP
YES
STOP

Appendix B. Output from CONSIZE for WB004C Vehicle

WEIGHT STATEMENT - LEVEL III

wb-004c, gr-ep lh2, rs-2100 - 25 klb p/l, 51.6 deg incl.

	WEIGHT (lb) LEVEL			CENTERS OF GRAVITY (ft./ft.)			MOM. OF INERTIA (slug-sq ft x10-6)		
	III	II	I	X/XREF	Y/YREF	Z/ZREF	XX	YY	ZZ
1.0 Wing			24563.	0.974	0.000	0.000	0.725	0.193	0.913
Exposed wing surface		18448.		0.994	0.000	0.000	0.704	0.087	0.788
Carry-through		3599.		0.974	0.000	0.000	0.015	0.004	0.018
Wing-body fairing		2516.		0.825	0.000	0.000	0.006	0.001	0.006
2.0 Tail			4041.	1.063	0.000	0.000	0.407	0.002	0.407
3.0 Body			71493.	0.633	0.000	0.000	0.337	9.280	9.379
LH2 tank		24156.		0.335	0.000	0.000	0.133	0.425	0.462
Structure	21047.			0.335	0.000	0.000	0.116	0.371	0.402
Insulation	3109.			0.335	0.000	0.000	0.017	0.055	0.059
LO2 tank		16326.		0.844	0.000	0.000	0.093	0.091	0.120
Structure	15100.			0.844	0.000	0.000	0.086	0.069	0.096
Insulation	1226.			0.844	0.000	0.000	0.007	0.022	0.023
Basic structure		21310.		0.738	0.000	0.000	0.091	2.868	2.891
Nose section	2683.			0.088	0.000	0.000	0.005	0.007	0.008
Intertank	8032.			0.651	0.000	0.000	0.059	0.030	0.047
Aft body/engine fairings	1988.			0.943	0.000	0.000	0.016	0.006	0.011
Thrust structure cone	8608.			0.974	0.000	0.000	0.011	0.008	0.008
Secondary structure		9701.		0.789	0.000	0.000	0.020	0.542	0.554
Crew cabin, work station	0.			0.648	0.000	0.097	0.000	0.000	0.000
P/L bay doors & hardware	1595.			0.651	0.000	0.000	0.002	0.010	0.010
P/L bay support str.	2000.			0.651	0.000	0.000	0.003	0.012	0.012
P/L container	2491.			0.651	0.000	0.000	0.003	0.015	0.015
Base closeout str.	600.			1.000	0.000	0.000	0.003	0.001	0.002
Body flap	2199.			1.041	0.000	0.000	0.007	0.000	0.008
Aft OMS/RCS pod str.	816.			0.987	0.000	0.000	0.003	0.000	0.003
4.0 Induced environment protection			29901.	0.603	0.000	0.000	0.219	3.020	3.090
TPS		27914.		0.601	0.000	0.000	0.219	2.883	2.954
Fuselage	18483.			0.422	0.000	0.000	0.145	0.056	0.103
Wing & fins	9431.			0.950	0.000	0.000	0.074	0.029	0.053
Internal insulation		1004.		0.520	0.000	0.000	0.000	0.091	0.091
Nose	233.			0.088	0.000	0.000	0.000	0.000	0.000
Payload bay doors	121.			0.651	0.000	0.000	0.000	0.000	0.000
Equipment bays	650.			0.651	0.000	0.000	0.000	0.000	0.000
Purge, vent, drn, & hazrd gas det		983.		0.750	0.000	0.000	0.000	0.000	0.000
5.0 Undercarriage and aux. systems			8727.	0.661	0.000	0.000	0.066	0.762	0.828
Nose gear		1356.		0.117	0.000	0.000	0.000	0.000	0.000
Running gear	257.			0.117	0.000	0.000	0.000	0.000	0.000
Structure	998.			0.117	0.000	0.000	0.000	0.000	0.000
Controls	100.			0.117	0.000	0.000	0.000	0.000	0.000
Main gear		7371.		0.761	0.000	0.000	0.066	0.000	0.066
Running gear	3153.			0.761	0.000	0.000	0.028	0.000	0.028
Structure	3802.			0.761	0.000	0.000	0.034	0.000	0.034
Controls	417.			0.761	0.000	0.000	0.004	0.000	0.004
6.0 Propulsion, main			69041.	0.988	0.000	0.000	0.139	1.267	1.325
Engines		47157.		1.040	0.000	0.000	0.113	0.028	0.080
Feed system		12779.		0.900	0.000	0.000	0.013	0.010	0.010
Pressurization system		862.		0.900	0.000	0.000	0.001	0.001	0.001
Gimbal actuation		3652.		0.975	0.000	0.000	0.009	0.002	0.006
Eng mounted heat shld		1623.		1.020	0.000	0.000	0.004	0.001	0.003
Helium pnuematic & purge system		2967.		0.566	0.000	0.000	0.000	0.000	0.000
7.0 Propulsion, reaction control (RCS)			4908.	0.654	0.000	0.000	0.022	0.812	0.818
Thrusters and supports		650.		0.902	0.000	0.000	0.003	0.158	0.159
Fwd	61.			0.088	0.000	0.000	0.000	0.008	0.008
Aft	589.			0.987	0.000	0.000	0.003	0.079	0.079
Propellant tanks		1731.		0.566	0.000	0.000	0.008	0.231	0.233
Distribution & recirculation		2526.		0.650	0.000	0.000	0.011	0.337	0.340
Lines, manifolds, & regulators	2043.			0.650	0.000	0.000	0.009	0.272	0.275
Valves	470.			0.650	0.000	0.000	0.002	0.063	0.063
Electric pumps	13.			0.650	0.000	0.000	0.000	0.002	0.002
Pressurization (included in OMS)		0.		0.566	0.000	0.000	0.000	0.000	0.000
8.0 Propulsion, orbital maneuver (OMS)			3168.	0.693	0.000	0.000	0.018	0.224	0.241
Engines		699.		0.987	0.000	0.000	0.004	0.000	0.004
Propellant tanks		975.		0.566	0.000	0.000	0.006	0.000	0.006

Pressurization and feed		1494.		0.639	0.000	0.000	0.008	0.095	0.103
Helium tanks	1234.			0.566	0.000	0.000	0.007	0.000	0.007
Lines (included in RCS)	0.			0.750	0.000	0.000	0.000	0.000	0.000
Valves	260.			0.987	0.000	0.000	0.001	0.035	0.035
9.0 Prime power			3256.	0.088	0.000	0.000	0.000	0.000	0.000
Fuel cell system		3256.		0.088	0.000	0.000	0.000	0.000	0.000
Cells	1820.			0.088	0.000	0.000	0.000	0.000	0.000
Reactant dewars	1436.			0.088	0.000	0.000	0.000	0.000	0.000
10.0 Electric conversion and distr.			8038.	0.450	0.000	0.000	0.042	0.826	0.857
Power conversion and distr.		1705.		0.088	0.000	0.000	0.000	0.000	0.000
Circuitry		4974.		0.519	0.000	0.000	0.014	0.342	0.346
Elect. pwr dist & cntrl	1465.			0.500	0.000	0.000	0.004	0.097	0.098
Avionic cabling	2434.			0.500	0.000	0.000	0.007	0.161	0.163
RCS cabling	132.			0.650	0.000	0.000	0.000	0.009	0.009
OMS cabling	211.			0.500	0.000	0.000	0.001	0.014	0.014
Connector plates	221.			0.600	0.000	0.000	0.001	0.015	0.015
Wire trays	511.			0.600	0.000	0.000	0.001	0.034	0.034
Electromech. act. (EMA) cabling		1359.		0.650	0.000	0.000	0.027	0.000	0.027
11.0 Hydraulic conversion and distr.			0.	0.000	0.000	0.000	0.000	0.000	0.000
12.0 Control surface actuation			3309.	1.024	0.000	0.000	0.074	0.004	0.078
Elevons		1427.		1.031	0.000	0.000	0.000	0.000	0.000
Tip fins		741.		1.063	0.000	0.000	0.074	0.000	0.074
Body flap		1141.		0.991	0.000	0.000	0.000	0.000	0.000
13.0 Avionics			1314.	0.243	0.000	0.000	0.002	0.206	0.207
Guid., nav., & contrl.		248.		0.088	0.000	0.000	0.001	0.022	0.022
Comm. & tracking		377.		0.088	0.000	0.000	0.000	0.000	0.000
Displays & contrl.		0.		0.088	0.000	0.000	0.000	0.000	0.000
Instrum. system		361.		0.651	0.000	0.000	0.001	0.030	0.030
Data processing		328.		0.088	0.000	0.000	0.000	0.021	0.021
14.0 Environmental control			2637.	0.295	0.000	0.000	0.007	0.182	0.180
Personnel system		0.		0.640	0.000	0.100	0.000	0.000	0.000
Equipment cooling		559.		0.088	0.000	0.000	0.000	0.000	0.000
Heat transport loop		1528.		0.325	0.000	0.000	0.007	0.060	0.058
Heat rejection system		551.		0.421	0.000	0.000	0.000	0.068	0.068
Radiators	326.			0.651	0.000	0.000	0.000	0.000	0.000
Flash evaporator system	225.			0.088	0.000	0.000	0.000	0.000	0.000
15.0 Personnel provisions			0.	0.000	0.000	0.000	0.000	0.000	0.000
Food, waste, & water mngmt.		0.		0.640	0.000	0.100	0.000	0.000	0.000
Seats		0.		0.640	0.000	0.100	0.000	0.000	0.000
16.0 Range safety			0.	0.000	0.000	0.000	0.000	0.000	0.000
17.0 Ballast			3328.	0.050	0.000	0.000	0.000	0.000	0.000
18.0 Payload provisions			0.	0.651	0.000	0.000	0.000	0.000	0.000
EMPTY			237723.	0.755	0.000	0.000	2.058	36.354	37.900
19.0 Growth allowance			35658.	0.750	0.000	0.000	0.321	1.462	1.605
EMPTY w/growth			273381.	0.754	0.000	0.000	2.379	37.817	39.506
20.0 Personnel			0.	0.000	0.000	0.000	0.000	0.000	0.000
Crew & gear		0.		0.640	0.000	0.100	0.000	0.000	0.000
Accessories		0.		0.640	0.000	0.100	0.000	0.000	0.000
21.0 Payload accomodations			0.	0.651	0.000	0.000	0.000	0.000	0.000
22.0 Payload			25000.	0.651	0.000	0.000	0.023	0.069	0.069
23.0 Residual and unusable fluids			6198.	0.553	0.000	0.000	0.012	0.552	0.557
Main prop. sys. pressurant		3211.		0.668	0.000	0.000	0.003	0.003	0.003
OMS and RCS		2128.		0.566	0.000	0.000	0.009	0.183	0.189
Subsystems		858.		0.088	0.000	0.000	0.000	0.000	0.000
25.0 Reserve fluids			9927.	0.737	0.000	0.000	0.009	0.604	0.613
Ascent		7927.		0.780	0.000	0.000	0.000	0.364	0.364
LH2	1003.			0.335	0.000	0.000	0.000	0.000	0.000
LO2	6924.			0.844	0.000	0.000	0.000	0.000	0.000
OMS		896.		0.566	0.000	0.000	0.005	0.000	0.005
RCS		1104.		0.566	0.000	0.000	0.003	0.124	0.127
26.0 Inflight losses			14548.	0.562	0.000	0.000	0.010	1.095	1.095
Vented ascent propellant		10373.		0.668	0.000	0.000	0.010	0.008	0.008
Fuel cell reactants		1612.		0.566	0.000	0.000	0.000	0.000	0.000
Evaporator water supply		2427.		0.088	0.000	0.000	0.000	0.000	0.000
Helium supply		136.		0.903	0.000	0.000	0.000	0.000	0.000
27.0 Propellant, main			2639372.	0.780	0.000	0.000	8.213	130.433	133.808
Start-up		37011.		0.780	0.000	0.000	0.115	1.829	1.876
LH2	4685.			0.335	0.000	0.000	0.015	0.057	0.061
LO2	32326.			0.844	0.000	0.000	0.100	0.073	0.116
Ascent		2602362.		0.780	0.000	0.000	8.098	128.604	131.932
LH2	329459.			0.335	0.000	0.000	1.048	3.986	4.289
LO2	2272903.			0.844	0.000	0.000	7.050	5.161	8.186
28.0 Propellant, reaction control			3988.	0.566	0.000	0.000	0.012	0.447	0.459
Orbital propellant		3000.		0.566	0.000	0.000	0.009	0.336	0.345
Entry propellant		988.		0.566	0.000	0.000	0.003	0.111	0.114
29.0 Propellant, orbital maneuver			24014.	0.566	0.000	0.000	0.144	0.000	0.144

100nmi alt. circularization prop.	2330.		0.566	0.000	0.000	0.014	0.000	0.014
220nmi alt. transfer/circ. prop.	10289.		0.566	0.000	0.000	0.062	0.000	0.062
Space station approach propellant	2392.		0.566	0.000	0.000	0.014	0.000	0.014
Deorbit propellant	9003.		0.566	0.000	0.000	0.054	0.000	0.054
PRELAUNCH GROSS		2996427.	0.773	0.000	0.000	10.803	175.422	180.656
		0.	0.000	0.000	0.000	0.000	0.000	0.000
Prelaunch gross		2996427.	0.773	0.000	0.000	10.803	175.422	180.656
Start-up losses		-37011.	0.780	0.000	0.000	-0.115	-1.829	-1.876
LH2	-4685.		0.335	0.000	0.000	-0.015	-0.057	-0.061
LO2	-32326.		0.844	0.000	0.000	-0.100	-0.073	-0.116
Gross lift-off		2959416.	0.773	0.000	0.000	10.687	173.590	178.777
Ascent propellant		-2602362.	0.780	0.000	0.000	-8.098	-128.604	-131.932
LH2	-329459.		0.335	0.000	0.000	-1.048	-3.986	-4.289
LO2	-2272903.		0.844	0.000	0.000	-7.050	-5.161	-8.186
Insertion (50X100 nmi orbit)		357055.	0.720	0.000	0.000	2.589	43.217	45.075
Ascent reserves		-7927.	0.780	0.000	0.000	0.000	-0.364	-0.364
LH2	-1003.		0.335	0.000	0.000	0.000	0.000	0.000
LO2	-6924.		0.844	0.000	0.000	0.000	0.000	0.000
OMS propellant - burn 1		-2331.	0.566	0.000	0.000	-0.014	0.000	-0.014
Insertion (100 nmi circular orbit)		346797.	0.720	0.000	0.000	2.575	42.719	44.564
Vented ascent propellant		-10373.	0.668	0.000	0.000	-0.010	-0.008	-0.008
OMS propellant - burns 2 & 3		-10289.	0.566	0.000	0.000	-0.062	0.000	-0.062
Insertion (220 nmi circular orbit)		326136.	0.727	0.000	0.000	2.503	42.252	44.035
OMS propellant - station approach		-2392.	0.566	0.000	0.000	-0.014	0.000	-0.014
RCS propellant		-3000.	0.566	0.000	0.000	-0.009	-0.336	-0.345
Payload delivered		-25000.	0.651	0.000	0.000	0.000	0.000	0.000
Payload accepted		25000.	0.651	0.000	0.000	0.000	0.000	0.000
Fuel cell reactants		-1612.	0.566	0.000	0.000	0.000	0.000	0.000
Evaporator water supply		-2427.	0.088	0.000	0.000	0.000	0.000	0.000
Helium supply		-136.	0.903	0.000	0.000	0.000	0.000	0.000
OMS propellant - deorbit		-9003.	0.566	0.000	0.000	-0.054	0.000	-0.054
Entry		307566.	0.740	0.000	0.000	2.426	39.573	41.279
RCS prop. (entry)		-988.	0.566	0.000	0.000	-0.003	-0.111	-0.114
Buoyancy		-8480.	0.488	0.000	0.000	-0.025	-0.950	-0.975
Landed		298099.	0.748	0.000	0.000	2.397	37.573	39.250
Payload (returned)		-25000.	0.651	0.000	0.000	-0.023	-0.069	-0.069
Landed (p/l out)		273099.	0.757	0.000	0.000	2.374	37.094	38.772
Payload accomodations		0.	0.651	0.000	0.000	0.000	0.000	0.000
Personnel		0.	0.000	0.000	0.000	0.000	0.000	0.000
Crew & gear	0.		0.640	0.000	0.100	0.000	0.000	0.000
Accessories	0.		0.640	0.000	0.100	0.000	0.000	0.000
Main prop. sys. pressurant		-3211.	0.668	0.000	0.000	-0.003	-0.003	-0.003
Subsystem residuals		-858.	0.088	0.000	0.000	0.000	0.000	0.000
Aux. propulsion residuals		-2128.	0.566	0.000	0.000	-0.009	-0.183	-0.189
OMS and RCS	-2128.		0.566	0.000	0.000	-0.009	-0.183	-0.189
Aux. propulsion reserves		-2000.	0.566	0.000	0.000	-0.009	-0.124	-0.132
OMS	-896.		0.566	0.000	0.000	-0.005	0.000	-0.005
RCS	-1104.		0.566	0.000	0.000	-0.003	-0.124	-0.127
Buoyancy		8480.	0.488	0.000	0.000	0.025	0.950	0.975
Empty w/growth		273381.	0.754	0.000	0.000	2.379	37.817	39.506
Landed - RTLS abort (max. p/l)		325274.	0.000	0.000	0.000	0.000	0.000	0.000

* INDICATES WEIGHT IS NOT WITHIN LIMITS OF WEIGHT EQUATION

wb-004c, gr-ep lh2, rs-2100 - 25 klb p/l, 51.6 deg incl.

DESIGN DATA

growth allowance fraction	0.1500
payload weight (lb)	25000.0000
additional down-payload (lb.)	25000.0000
payload bay diameter (ft.)	15.0000
payload bay length (ft.)	35.0000
payload volume (cu. ft.)	6185.0000
mission duration (days), max.	5.0000
oms delta v for tank sizing (ft./sec.)	1100.0000
oms delta v (ft./sec.) - burn 1	91.0000

oms delta v (ft./sec.) - burn 2	212.0000
oms delta v (ft./sec.) - burn 3	210.0000
oms delta v (ft./sec.) - station appr.	100.0000
oms delta v (ft./sec.) - deorbit	392.0000
lift-off t/w	1.2000
main eng. t/w (vacuum)	86.9200
main eng. isp (vacuum)	443.0000
thickness/chord	0.1000
aft dome to end of thrust str. (ft)	10.5000
ballast wt fraction	0.0140
nose area (ft^2)	2416.7896
body length (ft)	227.2194
body width (ft)	32.9758
body wetted area (ft^2)	21539.0313
body volume (ft^3)	168251.4687
intertank area w/o doors (ft^2)	4897.5361
aft skirt area (ft^2)	1790.7111
base heat shield area (ft^2)	205.1748
Lh2 tank wetted area (ft^2)	10871.0576
Lox tank wetted area (ft^2)	5284.7466
packaging efficiency	0.6637
wing-body fairing area (ft^2)	2515.9951
carry through width (ft)	32.9758
exposed wing root chord (ft)	56.4371
exposed wing taper ratio	0.3201
exposed wing span (ft)	79.9391
exposed wing aspect ratio	2.1459
exposed wing planform area (ft^2)	2977.8716
exposed wing wetted area (ft^2)	6170.1528
cos of sweep of exposed midChord	0.8872
tip fin planform area (ft^2)	542.3400
body flap planform area (ft^2)	614.1485
mass ratio	8.2884

SIZING PARAMETERS

Mass ratio	8.2884
Propellant mass fraction	0.8793
Body length (ft.)	227.2
Wing span (ft.)	112.9
Theoretical wing area (sq. ft.)	5099.9
Wing loading at design wt (psf)	63.8
Wing planform ratio, sexp/sref	0.58
Sensitivity of volume to burnout wt (cu. ft./klb.)	463.2
Burnout weight growth factor (lb/lb)	3.3

	BODY	WING
Total volume (cu. ft.)	168251.	13431.
Tank volume (cu. ft.)	111677.	0.
Fixed volume (cu. ft.)	0.	0.
Tank efficiency factor	0.6637	0.0000
Ullage volume fraction	0.0300	0.0300

PROPELLANT	FRACTION	DENSITY (lb/cu. ft.)	FLUID VOLUME (cu. ft.)	TANK VOLUME (cu. ft.)
lh2	0.1266	4.42	75825.	78170.
lox	0.8734	71.14	32501.	33507.
lox (Wing)	0.0000	71.14	0.	0.

Appendix C. I-DEAS, Version 6, Program File for Generation of Geometry and Finite Element Meshing of Fuel Tanks

```
C : units preference
K : $ return
K : $ mpos :; /O U U
K : inch
AP: 1 8 Change View
AP: 1 0 0 0 0
AP:  0.0            0.0            0.0
AP:  1.000000       0.0            0.0
AP:  0.0            1.000000       0.0
AP:  0.0            0.0            1.000000
AP:  0.1250000      0.2520000      0.2520000      15.00000
AP: -1.000000      -1.000000      -1.000000
AP:  1.000000       1.000000       1.000000
C : setup variables
C : use global symbol for component name
K : /options global_symbols on
K : enter comp_name
K : "lox_tank"
C :
K : #x_offset=424.515
K : #x_rotation=90.
K : #r1=191.84
K : #ecc1=sqrt(2)/2.
K : #barrel_len=987.168
K : #r2=191.84
K : #ecc2=sqrt(2)/2.
K : $ return
C : from lox_tank_zx.prg
C : below is born.prg type file
K : $ return
C : local switch off
K : $ /w gl:
K : $ mpos :; /ma na
K : LAB
K :
K : B
K : main
K : n
K : comp_name
K : ok
K : done
K : $ return
K : $ /cr ref cs
K : LAB
K : comp_name
K : cs
K : cs2
K :
K : tra
K : x_offset .0 .0
K : rot
K : x_rotation .0 .0
K : done
C : attach to part, coord system, plane
K : $ /w at
K : LAB
K : comp_name
K : coordinate
K : cs3
K : xy_plane
```

```
K : $ mpos :; /v wp
K : $ return
C : create fwd ellipse
K : $ /cr el co
K : OP
K :  FX -r1*ecc1
K :  FY .0
K :  SX .0
K :  SY r1
K :  TX -r1*.5
K :  TY r1*.707
K :  RO .41
K : OKAY
P :
K : $ return
C : create aft ellipse
K : $ /cr el co
K : OP
K :  FX barrel_len+r2*ecc2
K :  FY 0
K :  SX barrel_len
K :  SY r2
K :  TX barrel_len+r2*.5
K :  TY r2*ecc2
K :  RO .41
K : OKAY
C : Create tangent line to tops of fwd and aft domes
K : $ return
K : $ /cr l si
K : OP
K : FX .0 FY r1
K : SX barrel_len SY r2
K : OKAY
K : $ return
K : $ /cr se
K : OP
K : AU
K : Y
K : ST
K : N
K : Okay
K : LAB
K : comp_name
K : curve
K : 2
K : done
K : RE
K : label
K : comp_name
K : section
K : 1
K : KEY
K : 10 0 0
K : angle
K : -180
K : ok
K : $ return
P : start simulation
K : $ $ $ /ta xx SI
K : $ $ $ /ta ME
C : start groups
```

```
K : $ mpos :; /GR IP DI
K : F
K : fem_one
K : ok
K : $ return
K : /
K : group create
K : label
K :
K : surface
K : join5
K : 3
K : done
K : fwd_dome
K : /gr cr
K : label
K :
K : surface
K : join5
K : 2
K : done
K : barrel
K : /gr
K : cr
K : label
K :
K : surface
K : join5
K : 1
K : done
K : aft_dome
C : mesh aft dome
K : DFN
K : SH
K : LAB
K :
K : 5
K : 1
K : done
K : MT
K : MA
K : MO
K : DC
K : LAB
K :
K : 5
K : 1
K : DFE
K : 10
K : set
K :
K :
K : !
K : !
K : !
K : 20
K : done
K : E
K :
K : 5
K : *
```

```
K :
K : 5
K : 1
K :
K :
K : $
K : DEL
K :
K : 5
K : *
K : C
K : $
K : vie redi; don
K : DFE
K : !
K : FU
K :
K :
K :
K :
K : $
K : DFE
K : !
K : Canc
K : PM
K : PM
K : Okay
C : from domemesh file worked ok
C : aft dome
K : define
K : SH
K : label
K :
K : join5
K : 1
K : done
K :  MT MA
K :  MO
K :  DC
K : LAB
K :
K : 5
K : 1
K :  DFE
K : 10
K :
K : 10
K :
K : CANC
K :  PM
K : OKAY
C : barrel
K : define
K : sh
K : label
K :
K : join5
K : 2
K : done
K :  MT MA
K :  MO
```

```
K :  DFE
K : 12
K :
K : 10
K : CANC
K :  PM
K : OKAY
C : fwd dome
K : define
K : sh
K : label
K :
K : join5
K : 3
K : done
K :  MT MA
K :  MO
K :  DC
K : LAB
K :
K : 5
K : 6
K :  DFE
C : was 5
K :10
K :
K : CANC
K :  PM
K : OKAY
C :
K : /group
K : set_current
K : fwd_dome
K : Display_group
C : used to add entities to a group in this case
C : related elements added to a surface
K : /group
K : set_current
C : (variable) name of group
K : aft_dome
K : Display_group
K : add
K : related_to
K : ELEM
K : LAB
K :
K : label
K : filter
K : pickable_menu
K : 3
K : done
K : pick_only
K :
K : surface
K : join5
C : (variable) surface label to add elements to
K : 1
K : done
K : done
K : DG
C : used to add entities to a group in this case
```

```
C : related elements added to a surface
K : /group
K : set_current
C : (variable) name of group
K : fwd_dome
K : Display_group
K : add
K : related_to
K : ELEM
K : LAB
K :
K : label
K : filter
K : pickable_menu
K : 3
K : done
K : pick_only
K :
K : surface
K : join5
C : (variable) surface label to add elements to
K : 3
K : done
K : done
K : DG
C : used to add entities to a group in this case
C : related elements added to a surface
K : /group
K : set_current
C : (variable) name of group
K : barrel
K : Display_group
K : add
K : related_to
K : ELEM
K : LAB
K :
K : label
K : filter
K : pickable_menu
K : 3
K : done
K : pick_only
K :
K : surface
K : join5
C : (variable) surface label to add elements to
K : 2
K : done
K : done
K : DG
```

Appendix D. Listing of I-DEAS Finite Element Property Assignment Program

```
C : Zoran's Program for property assignment
K : # elno=1
K : # elmax=1
K : # nth=5
K : # tinc=.00
K : # thk=.001
K : # prefix=" "
K : #input "starting element no. =>" elno
K : #input "ending element no. =>" elmax
K : #input "how many elements per property car =>" nth
K : # nthmax=nth+1
K : #input "property prefix string =>" prefix
C :  #input "starting thickness minus .001 =>" thk
C : First property setup
K : #   if (0 EQ 0) then; ,
K : #     pname=prefix+elno; ,
K : #     thk=thk+tinc; ,
K : #     iter=0
K :       /ph cr; ,
C :       cr; ,
K :       tn; ,
K :       pname; ,
K :       no; ,
K :       tk; ,
K :       thk; thk; thk; thk; ,
K :       done
K :       DES
C :  first pause
C : Loop over all elements
K : # loop1:
K : # iter=iter+1
K : #   output "iter is " iter
C : New property if at right increment
K : #   if (iter GT nth) then; ,
K : #     pname=prefix+elno; ,
K : #     thk=thk+tinc; ,
K : #     iter=1; ,
K :       /ph cr; ,
K :       thin_shell; ,
K :       pname; ,
K :       no; ,
K :       tk; ,
K :       thk thk thk thk; ,
K :       done
K :       DES
C :
K :     /element
K :     modify
K :     label
K :     elno
C : # ON_ERROR IGNORE
C : # ON_ERROR GOTO skip1
K :     done
K :     P
K :     pname
K :     yes
K : # output "elno prop modified"
K : # skip1:
```

```
K : #   elno=elno+1
K : # if (elno LE elmax) then #GOTO loop1
E : **** END OF SESSION ****
```

Appendix E. Listing of CONSIZE_MOD.txt file

```
From   I:\Zoran\VAB\Son of HAVOC\wb004c.xls

      WEIGHT      STATEMENT   -     LEVEL III

      wb004c      external    "p/l,"      ssme  block2      -

CONSIZ_Component  FEA_GROUP   Weight      Mapping     XB    XE
begin_components
wing_exposed      wing_exposed      18448 component   0     0
wing_carrythru    wing_carrythru    3599  component   0     0
wing_fairing      wing_fairing      2516  component   0     0
vert_fin    vertical_tail     4041  component   0     0
nose_assy   nose_assy   2683  component   0     0
LOX_tank    lox_tank    15100 component   0     0
LOX_cryo_insul    lox_tank    1226  component   0     0
intertank_assy    intertank_assy    8032  component   0     0
LH2_tank    lh2_tank    21047 component   0     0
LH2_cryo_insul    lh2_tank    3109  component   0     0
payload_bay fuselage_side     6086  fofx  1574  1974
thrust_str  thrust_str  8608  component   0     0
aft_bd_eng_fair   aft_bd_fairing    1988  component   0     0
aft_oms_pod aft_bd_fairing    816   fofx  2630  2730
base_closeout     base_closeout     600   component   0     0
body_flap_assy    body_flap_assy    2199  component   0     0
tps_fuselage      nose_fus_thrust   18483 component   0     0
tps_wing_fin      tps_wing_fin      9431  component   0     0
insulation_nose   nose_assy   233   component   0     0
insulation_pl_door      fuselage_top      121   fofx  1574  1974
insulation_equip  fuselage_top      650   fofx  1574  1974
purge_vent_drain  fuselage_side     983   fofx  500   2730
nose_gear   nose_bot    1356  fofx  240   400
main_gear   main_gear   7371  component   0     0
main_engines      thrust_str  69041 fofx  2600  2700
rcs_system_fwd    fuselage_side     4257  fofx  1500  1880
rcs_system_aft    thrust_str  650   fofx  2460  2470
oms   fuselage_side     3168  fofx  1860  1970
prime_pwr   nose_assy   3256  fofx  200   280
elec_conv_dist    nose_assy   1705  fofx  200   280
elec_circ   fuselage_side     4974  fofx  1200  1600
elec_cabl   fuselage_side     1359  fofx  1600  1970
cs_actuation_el   wing_rear_spar    1427  component   0     0
cs_act_fin  fin_rear_spar     741   component   0     0
cs_actuation_bf   bodyflap_spar     1141  component   0     0
avionics    fuselage_side     1314  fofx  560   760
env_ctrl    fuselage_side     2637  fofx  560   1010
ballast     nose_assy   3328  fofx  0     180
growth_conting    nose_fus_thrust   35658 component   0     0
payload     intertank_assy    25000 component   0     0
residual_fluids   fuselage    6198  fofx  1000  2000
reserve_fluids    fuselage    9927  fofx  2000  2100
inflight_losses   fuselage    14548 fofx  1000  2000
propellant_rcs    fuselage_side     3988  fofx  1500  1600
propellant_oms    fuselage_side     24014 fofx  1500  1600
end_components
      empty 237724      338716      100992

      sum   357057      536281      179224      ok
```

Appendix F. Listing of Commands to run JAVA Programs

```
# alias consiz2unv java -classpath ~/javacode/public consiz2unv  junk.unv  tsto_consiz.in.txt masses.prg
alias consiz2unv java -classpath ~cerro/javacode/public consiz2unv  $1 $2 $3

# java -classpath ~cerro/javacode/public:. combine_loadsets  input_filename  output_unvfile_name
alias combine_loadsets java -classpath ~cerro/javacode/public:. combine_loadsets  $1  $2
```

Appendix G. Typical Tank Head Pressure I-DEAS V6 Program File

```
C : Start USER INPUT __________________________
C : position positive point on neg side to reverse pressure sign
K : /options global_symbols on
K : enter ldst_name
K : yes
K : "lh2_40%_2.93_0_0.17"
K : /options global_symbols on
K : enter grp_name
K : yes
K : "lh2_tank_elements"
K : # ullage=.0
K : # rho_g=.0075042
C : Vehicle acceleration
K : # ax=-2.93
K : # ay=.0
K : # az=-0.17
K : # declare pos(3)
K : # pos(1)=1441.
K : # pos(2)=.0
K : # pos(3)=.0
C : END USER INPUT __________________________
C :
K : # declare pos2(3)
K : # pos2(1)=pos(1)-100.
K : # pos2(2)=pos(2)
K : # pos2(3)=pos(3)
K : # ax=-ax
K : # ay=-ay
K : # az=-az
K : $ return
K : $ mpos :; /O P
K :  P 1;
K :  ME ON
K : OKAY
K :  P 2;
K :  FD OF
K : OKAY
K : OKAY
K : $ return
K : /ta bo
K : SE
K : ST
K : LOAD
K : SE
K : ldst_name
K : CR
K : make_current
K : Canc
K : CR
K : AD
K : H
K : UG
K : directory
C : next
C : backup
K : grp_name
K : done
```

```
K : POI
K : KEY
K : ax ay az
K : KEY
K : pos
K : KEY
K : pos2
K : ullage
K : rho_g
K : $ return
K : $ mpos :; /O P
K : U
K : Y
K : Okay
K : $ return
```

Appendix H. Calculation of Dynamic Thrust Factor for Liftoff Condition

Based on Dutch Mayer's Internal Memo

List of Symbols:
W = Vehicle weight at liftoff
Ta = Axial component of the thrust force at liftoff
Tn = Normal component of the thrust force at liftoff

Before engines ar started, the weight **W** of the vehicle is supported solely at the eight tiedown mounts. Figure G1 is a simplified cartoon representation of the vehicle that shows only one tie-down mount on the bottom left of the vehicle and one engine mount on the right. There are actually six engine mounts in the full vehicle.

Define thrust to weight ratios: **Ta/W = ta**, and **Tn/W = tn**. Then **Ta = ta W**, and **Tn = tn W**.

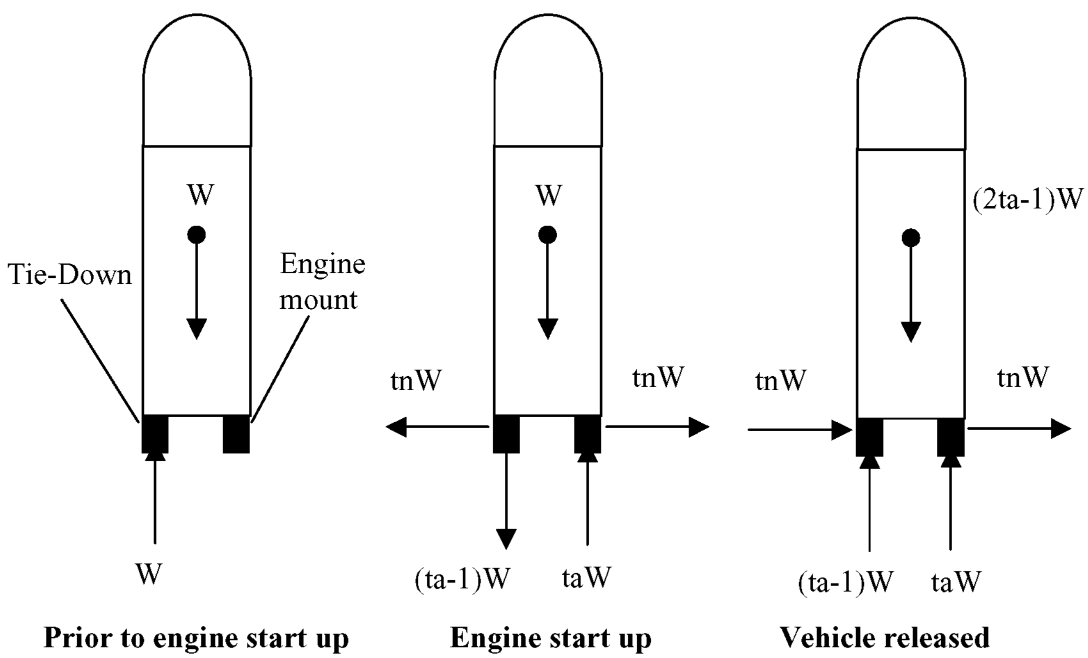


Figure G1. Free Body Diagram of a Launch Vehicle During Liftoff

The engines are then lit, and supply **ta W** axial thrust and **tn W** normal thrust at the engine mount. The second free body diagram in Figure G1 shows the "quasi" static condition. The **ta W** axial thrust is supported by the weight **W** of the vehicle and and a reaction **(ta-1) W** at the tie-downs. The transverse thrust tn **W** is supported solely at the eight tie-down mounts.

Finally, the tie-down bolts are relised and the reaction forces "spring back". Because of this dynamic phenomena, there is now **(ta-1) W + ta W = (2ta-1) W** axial thrust and **2tnW** normal thrust. This equals to a **(2ta-1) W/(ta W) = 2-(1/ta)** axial dynamic factor, and **2tn W/(tn W) = 2** normal dynamic factor.

REPORT DOCUMENTATION PAGE	*Form Approved* *OMB No. 0704-0188*

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	September 2002	Technical Memorandum

4. TITLE AND SUBTITLE
A Procedure for Structural Weight Estimation of Single Stage to Orbit Launch Vehicles (Interim User's Manual)

5. FUNDING NUMBERS
706-88-21-03

6. AUTHOR(S)
Zoran N. Martinovic and Jeffrey A. Cerro

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
NASA Langley Research Center
Hampton, VA 23681-2199

8. PERFORMING ORGANIZATION REPORT NUMBER
L-18169

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA/TM-2002-211931

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified-Unlimited
Subject Category 15 Distribution: Nonstandard
Availability: NASA CASI (301) 621-0390

12b. DISTRIBUTION CODE

13. ABSTRACT (*Maximum 200 words*)

This is an interim user's manual for current procedures used in the Vehicle Analysis Branch at NASA Langley Research Center, Hampton, Virginia, for launch vehicle structural subsystem weight estimation based on finite element modeling and structural analysis. The process is intended to complement traditional methods of conceptual and early preliminary structural design such as the application of empirical weight estimation or application of classical engineering design equations and criteria on one dimensional "line" models. Functions of two commercially available software codes are coupled together. Vehicle modeling and analysis are done using SDRC/I-DEAS, and structural sizing is performed with the Collier Research Corp. HyperSizer program.

14. SUBJECT TERMS
Weight Estimation, Structural Design, Structural Sizing, Reusable Launch Vehicle Design, Finite Element Modeling

15. NUMBER OF PAGES
59

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	UL