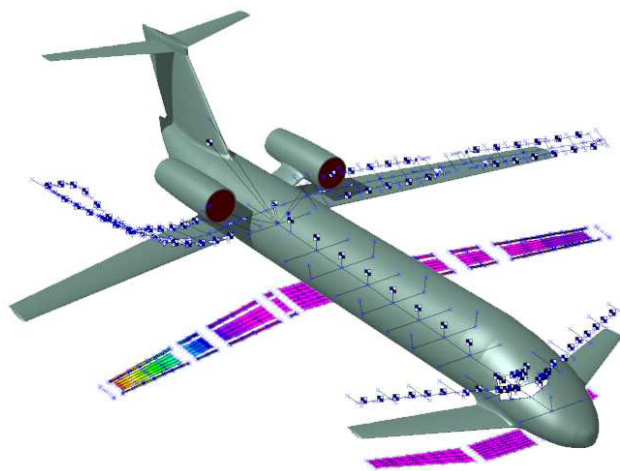


NeoCASS

Next generation Conceptual Aero Structural Sizing



Dipartimento di Scienze e Tecnologie Aerospaziali
Politecnico di Milano



Report name	NeoCASS Next generation Conceptual Aero Structural Sizing
Date	December 11, 2013
Partner	Dipartimento di Scienze e Tecnologie Aerospaziali Politecnico di Milano
Author	L. Cavagna, S. Ricci
Delivery number	D.XXXX
Dissemination level	Public

Contents

A FEMGEN: Finite Element Model GENerator	2
A.1 FEMGEN program	4
A.1.1 Geometry definition	5
A.1.2 Mesh parameters	7
A.1.3 Properties	8
A.1.4 FEMGEN and SMARTCAD coupling	10
A.2 Testcase	16
A.3 Conclusions	17

List of Figures

A.1 FEMGEN layout.	3
A.2 FEMGEN layout.	4
A.3 Wing geometry.	6
A.4 Wing box spar layout, 0.2 and 0.6 chord percentage at each control section.	6
A.5 Mesh nodes.	7
A.6 Wing box spar layout, 0.2 and 0.6 chord percentage at each control section.	8
A.7 FE mesh.	9
A.8 FE mesh internal view.	10
A.9 Mesh nodes and aerodynamic mesh.	11
A.10 Structural properties interpolation.	12
A.11 SMARTCAD structural model of the wing.	16
A.12 Vibration modes computed in SMARTCAD (magnified by 50).	18
A.13 Vibration modes.	19
A.14 Deformed shape for static pull-up, $n_z = 2.5$	19

List of Tables

A.1	Inertial properties for wing-box only.	17
A.2	Inertial properties for wing-box and lumped masses.	17
A.3	Comparison for vibration modes.	17

Appendix A

FEMGEN: Finite Element Model GENerator

The models available within NeoCASS are relatively simple and rely on:

- beam finite elements;
- bi-symmetric wing box with rectangular section;
- assumed structural concepts having a defined set of design variables which concur to the definition of the stiffness and structural mass for each section. The value of the design variables are determined through a structural optimization process resulting in a minimum weight layout[2] or through design principles of minimum weight [1, 3];
- assumed distribution of secondary structural mass along fuselage axis and along leading and trailing edge of lifting surfaces;
- user defined distribution of fuel in wing-tanks and payload in fuselage.

FEMGEN is placed downstream of NeoCASS procedures and focuses on the finite element analysis of the wing-box. Fig. A.1 shows a sketch of the data flow between FEMGEN and NeoCASS:

- the `.xml` file is the starting point: all the necessary geometry and structural parameters are provided;
- GUESS sizes the aircraft starting from `.xml` file and exports a stick model and aerodynamic mesh to SMARTCAD for numerical assessment;
- FEMGEN is finally provided with wing geometry from `.xml` file, stiffness distribution from GUESS and non-structural masses and loads from SMARTCAD .
- user-inputs are provided to FEMGEN through its interactive interface.

A Finite Element (FE) model of the wing-box for NASTRAN[®] is generated by FEMGEN. The model is given as a collection of text files.

Different kind of analysis can be run (see [4] for further information):

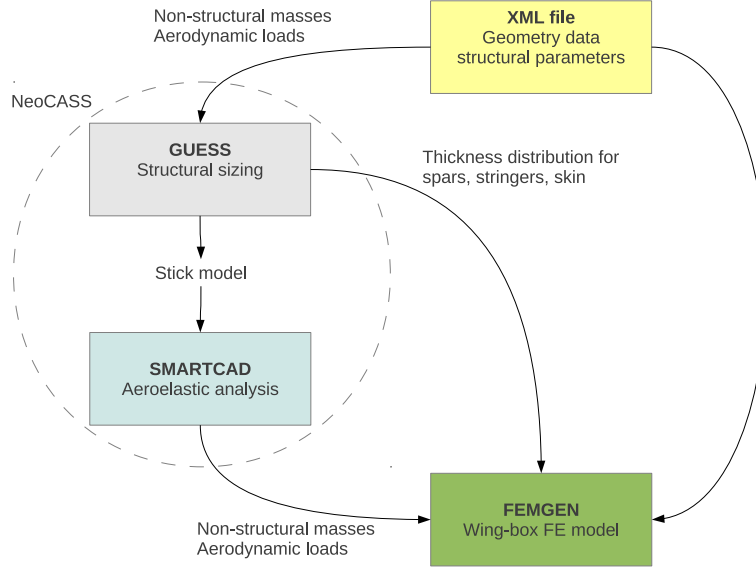


Figure A.1: FEMGEN layout.

- static analysis (SOL 101);
- eigenvalue analysis (SOL 103);
- flutter analysis (SOL 145);
- optimization and sensitivity (SOL 200).

Starting from the stick model sized by GUESS , the FE model is made-up of:

- shell elements for spars, skin and ribs;
- beam elements for stringers and spar caps.

Constraints at symmetry plane and wing-fuselage intersection are included, together with the aerodynamic model necessary for flutter assessment (SOL 145). For each bay, i.e. the segment of wing-box between two consecutive ribs, the following independent properties are defined:

- fore and rear spar (shell thickness);
- upper and lower skin (shell thickness);
- upper and lower stringers (beam area);
- upper fore/rear spar caps and lower fore/rear spar caps (beam area).

Thus, the total number of properties for each bay is 10.

Spar caps and stringers have pre-defined cross section in shape (L and Z respectively) and proportions (see the source file `pbeam_generator.m`). This simplifies the creation of the model and allows the user to define his own sections. Fig. A.2 shows the default cross sections for spar caps and stringers respectively.

FEMGEN determines the value of bay properties from GUESS in two different ways. Either method is chosen according to the value of variable *kcon* given in the `xml` file:

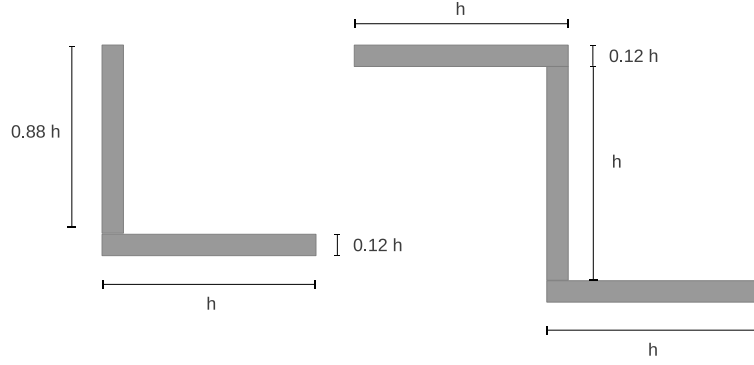


Figure A.2: FEMGEN layout.

1. when $kcon = 9$, the thickness of spar and skin and the area of stringers is directly available; an interpolation along wing axis is hence carried out;
2. for all other cases, a minimization process is solved for each section to determine the value of skin thickness, spar thickness and stringer area which best approximate the values of total area, second moments of inertia I_{xx} and I_{yy} and torsional stiffness J determined by GUESS .

When creating the FE model, ribs are defined directly by the user in terms of thickness and spanwise location. Indeed, ribs are usually not considered by GUESS . The only case is for $kcon = 9$ which has one dedicated parameter (`rpitch` in `xml` file) for spanwise rib pitch. This length affects stringer and skin critical buckling load used as limit during the sizing process. No other effect is considered; i.e. rib weight and stiffness is totally neglected by GUESS .

A.1 FEMGEN program

FEMGEN is a collection of routines for geometry, meshing and data output. The program is started by typing at the prompt:

`wing_fem(output_filename, xml_filename, input_filename)` where:

- `output_filename` is a string with the output filename head to create;
- `xml_filename` is a string with the input `xml` file of the aircraft.
- `input_filename` is a string with the input `.m` file for FEMGEN. This file is not mandatory. if an empty input is given, FEMGEN will run its own interactive user-interface to gather all the data required. When the process is ended a `.m` file will be available with a list of all the parameters used. The user can modify this file and provide it as input to FEMGEN to directly create a new set of NASTRAN files.

For example, `wing_fem('test','B747-100.xml')` will create a different NASTRAN main files with headname `test` such as `test_101.dat`, `test_145.dat`, `test_200.dat`, for the wing defined in the file `B747-100.xml`. An input file `test.m` will be created as well for successive runs.

The program has an interactive input interface to provide all the required parameters:

- geometry layout of the wing-box, i.e. spar position, ribs;
- meshing parameters, i.e. number of elements along spars, between ribs, along ribs;
- different settings for the type of analysis to run, i.e. design variables link (SOL 200), aerodynamic mesh (SOL 145).

The main launch files for each NASTRAN solver are automatically created. The most typical settings are already included. Freedom is left to the user to modify the main files and the secondary files with FE properties, e.g. number of modes to extract, material properties and type of constraints. A complete list of secondary files is given in Sec. [A.1.3](#).

A.1.1 Geometry definition

The first section of FEMGEN deals with geometry inputs to define the layout of the wing-box. In the following, ATR-72 aircraft will be considered as a test case to show the application of FEMGEN to a real case.

Spars, ribs, stringers

The first section of the program is focused on the main geometry input data and wing box layout. The following data are recovered through the user-interface:

- spars position;
- ribs orientation;
- stringers chordwise layout.

Fig. [A.3](#) shows the image opened on the screen when the program is run. Control sections (green lines) are selected as aid in defining the wing-box. The wing planform is made-up at most of five patches (trapezoids delimited by two blue lines and two green lines) connecting the control sections:

- carry-through from section 1 to 2;
- inboard from section 2 to 3;
- midboard from section 3 to 4;
- outboard from section 4 to 5;
- winglet from section 5 to 6 (not considered in this example).

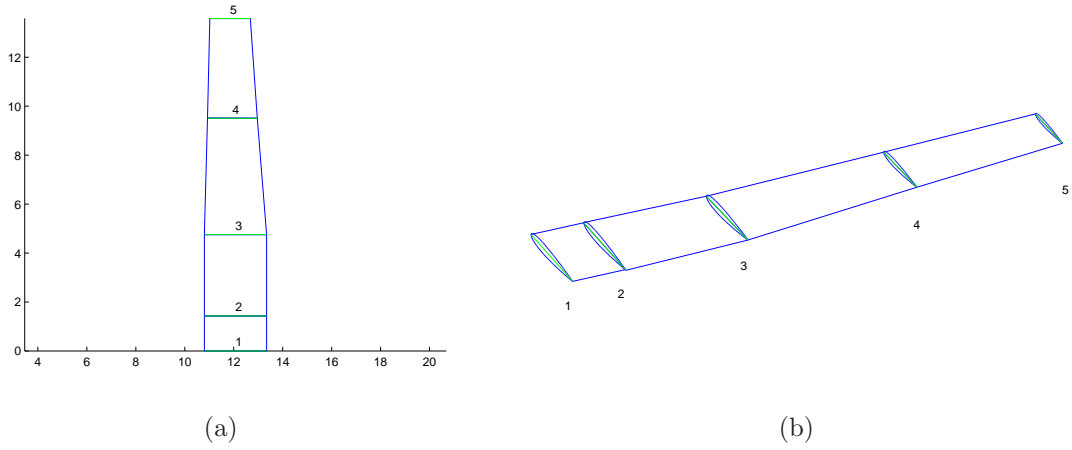


Figure A.3: Wing geometry.

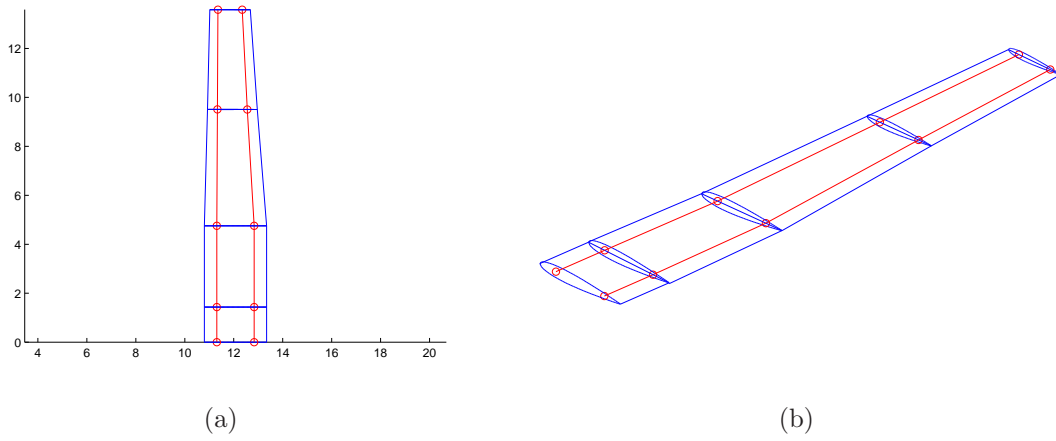


Figure A.4: Wing box spar layout, 0.2 and 0.6 chord percentage at each control section.

Wing planform and airfoils at control sections are recovered from the `xml` file. To define the wing-box limits, FEMGEN requires the user to input the chord fraction along the control section for fore and aft spar. These points are reported as red dots in Fig. A.4. When connected along wing-span, spars (red lines) are settled. Two options are available for ribs alignment:

1. wind, ribs are along free-stream;
2. wing-box axis, ribs are orthogonal to spars.

In the first case, ribs are normal to y -axis, thus are laid along x -axis. In the second case, ribs are normal to the axis given by ribs midpoints. Two options are available for stringers as well:

1. fixed number;
2. fixed pitch, i.e. chordwise distance between two consecutive elements.

In the first case, the same number of stringers will be laid along wing-span. In the second case the number of stringer will vary for each wing patch. FEMGEN requires one value

for upper and lower stringers respectively. At this point, the layout of the wing box is completely defined and a second phase starts to gather all the parameters ruling the meshing process.

A.1.2 Mesh parameters

The mesh section allows the user to rule the discretization of the FE model. FEMGEN requires the following parameters:

- number of elements between two consecutive ribs;
- number of elements between two consecutive stringers;
- number of elements along wing box height for fore and aft spar respectively;
- number of radial elements on ribs.

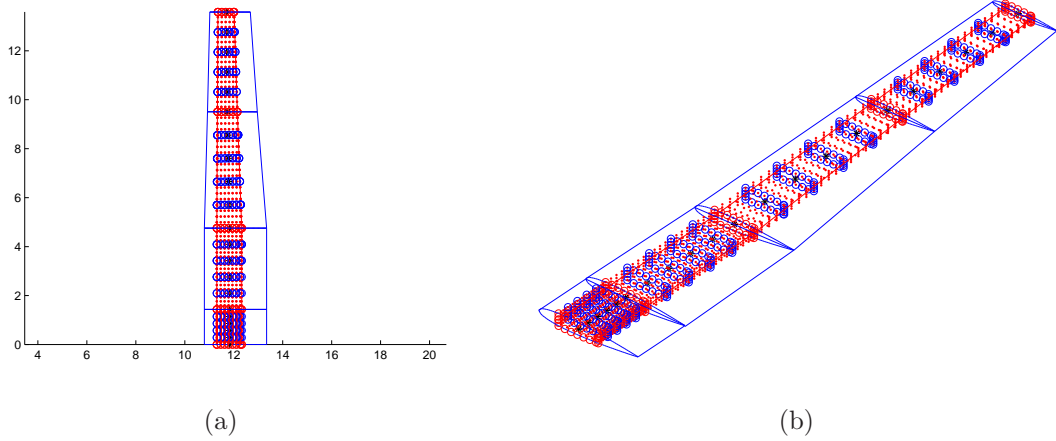


Figure A.5: Mesh nodes.

Fig. A.5 shows the nodes of the FE mesh. The blue ones belong to the user-define ribs. The red ones belong to stringers and skin. In this example each wing patch has four internal ribs. Fig. A.6 shows the FE model in detail for the testcase considered. In this simple case, 4 elements are laid along fore and aft spar and between two consecutive ribs. Also, 4 radial elements appear on each rib. The elements start at the rib center and connect the nodes along the skin and the spars. The skin has 6 chordwise elements, and hence 7 stringers. The white squares along the rib planes are lumped non-structural masses (see Sec.A.1.4). Fig. A.7 and A.8 shows further details of the FE model. In particular, Fig. A.8 also shows the reference frames created for control surface definition. Each frame has a local y axis aligned with control surface hinge line and is located at the hinge origin. At the end of the process, the user defines the aerodynamic DLM mesh to create. For each wing patch the following parameters are required:

- number of spanwise panel;
- number of chordwise panels;

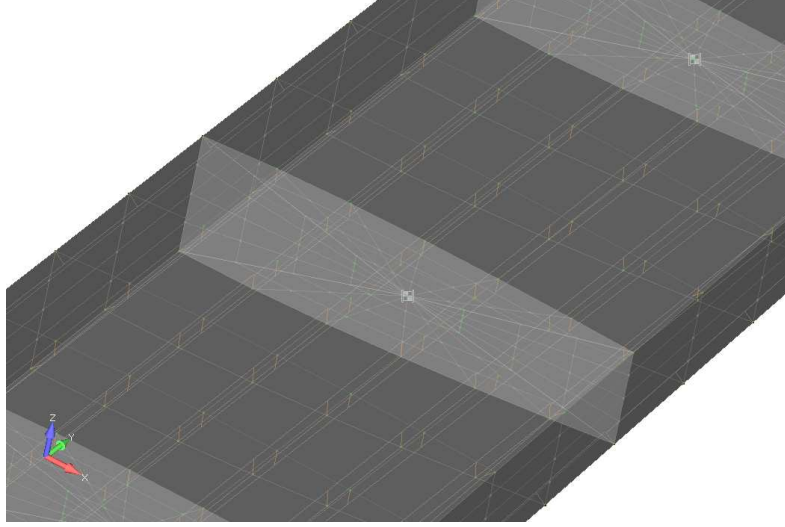


Figure A.6: Wing box spar layout, 0.2 and 0.6 chord percentage at each control section.

- number of chorwise panels along control surfaces.

Fig. A.9 shows the aerodynamic mesh created for the testcase considered. Control surfaces parameters, e.g. chord and span fractions, are directly recovered from the `xml` file. Moreover, when a control does not extend over the whole patch span, additional patches are added and the user is required to provide data for them as well. For the testcase considered, an additional patch is added at the left of the aileron. FEMGEN creates also all the cards required for NASTRAN to run aeroelastic flutter analysis (SOL 145):

- CAERO1, AESURF, AELIST and CORD2R cards with aerodynamic mesh geometry, control surface definition and their hinge axis;
- SPLINE1, CORD2R cards for spatial coupling to FE mesh through NASTRAN beam splines.

A.1.3 Properties

Two types of material are available (see the file `pbeam_generator.m`):

1. aluminum;
2. carbon-fiber.

The user can modify the file `mat.dat` to change the default properties. The carbon fiber material is supposed to be symmetric and balanced. In this respect, a MAT8 card is used. FEMGEN needs to recover the properties for the spar, stringers and skin to define the set of properties for each bay. The user can either provide the output file from GUESS (`_guess.mat`) or set each item to a constant value along span. When a GUESS file is given, an interpolation or a minimization process can be carried out depending on the value of the parameter `kcon` in the `xml` file. As mentioned above, when $kcon = 9$ the properties are interpolated from GUESS database. For all other cases, a minimization problem is solver for each section. Figure A.10 shows the results of the interpolation

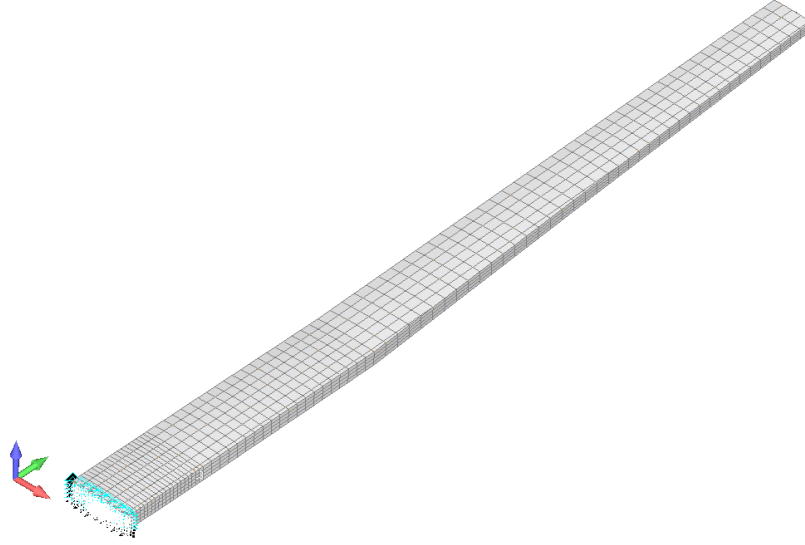


Figure A.7: FE mesh.

process for skin and web thickness and stringer area. FEMGEN creates a different set of properties for each bay. However, considering NASTRAN optimization runs (SOL 200), design variables may be mutually linked for computational savings or physical issues, e.g. avoid unwanted and unreal excessive spanwise tapering of elements. Thus FEMGEN asks the user to define if each bay property is independent or not. For each wing patch (at most 5 as reported above), the user specifies how many independent sets to create and which bay property set belongs to them.

Each wing patch has a number of bays which depends on the number of ribs given. For the example considered, the user supplies 4 ribs for the carrythrough patch, resulting in a total number of bays of 5. The property index goes from 0 to 4 in this case. All the information will be displayed on the screen and the user can simply define each set. Let us consider two examples. The user can simply create one set as:

1. $\{0,1,2,3,4\}$;

or create 3 sets as:

1. $\{0,1\}$;
2. $\{2,3\}$;
3. $\{4\}$.

Sets definition will affect the final list of NASTRAN DESVARs to be used during the optimization/sensitivity design (SOL 200). At the end of FEMGEN, the following files are available:

- `output_filename_101.dat`, `output_filename_145.dat` and `output_filename_200.dat`
main file for SOL 101, 145 and 200;
- `mat.dat` with material properties;

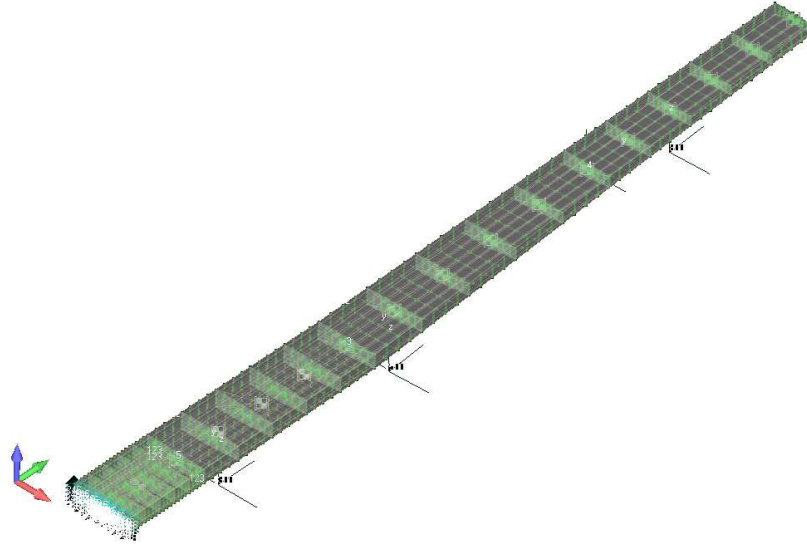


Figure A.8: FE mesh internal view.

- `grid.dat` with nodes along spars and skin;
- `element.dat` with skin and spar plates definition;
- `pshell.dat` with skin and spar plate properties;
- `beam.dat` with spar cap and stringers beam elements;
- `rib.dat` with rib nodes, plates and properties;
- `spc_sym.dat` with constraints at symmetry plane (T2,T4,T6);
- `spc_fuse.dat` with pins at wing fuselage intersection (T1,T2,T3);
- `caero.dat` with DLM model (SOL 145 only);
- `desvar.dat`, `dresp1.dat`, `dtable.dat`, `dvp1rel1.dat`, `dvp1rel2.dat`, `analytical_buckling.dat` with optimization cards (SOL 200 only);

The constraints at wing-fuselage intersection by default are represented by three pins. Two pins are located at the front spar and the last one at the rear spar. A set of `.mat` and `.fig` files are created as well. The former is a backup of internal data. The latter is a set of figures of the wing-box.

The file `ribs_data.mat` must not be deleted because it is required in a later stage (see the following section).

A.1.4 FEMGEN and SMARTCAD coupling

At the end of the process, the FE model of the wing-box is defined in terms of grids, elements and properties.

However, two points are lacking in order to have a complete model ready to run:

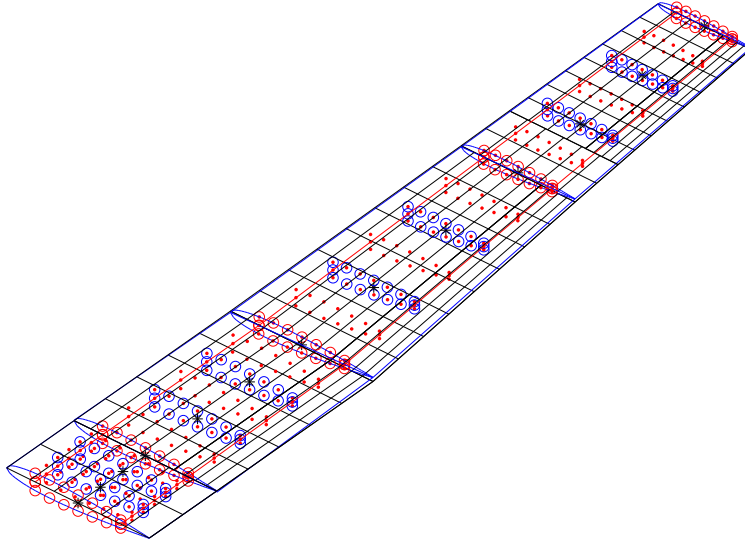


Figure A.9: Mesh nodes and aerodynamic mesh.

- the effects of secondary structural mass and non-structural mass in inertial loads;
- external loads.

FEMGEN allows the coupling with a SMARTCAD model to introduce the two missing sets of data, as reported in Fig. A.1.

The secondary structural mass is estimated by GUESS through statistical methods starting from the weight of the wing-box. These terms are exported to SMARTCAD main file as lumped masses along wing leading and trailing edge. as well as non structural masses such as engines. They appear in the main file which represents the aircraft in Max Operative Empty Weight (MOEW) configuration. Referring to Fig.A.11, engines and secondary masses as represented as dots linked through a black dashed line to the stick model. On the other hand, the payload distributed along wing-box such as fuel is exported in separate files according to the tank filling-up status.

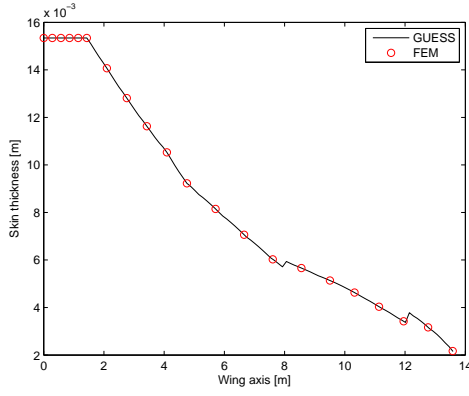
To transfer data from SMARTCAD to FEMGEN in terms of secondary structural masses, fuel distribution and maneuver loads, the user takes the main SMARTCAD file and includes:

- a mass configuration file (files created by GUESS with a CONM_CONF appended)
- a file with maneuvers defined.

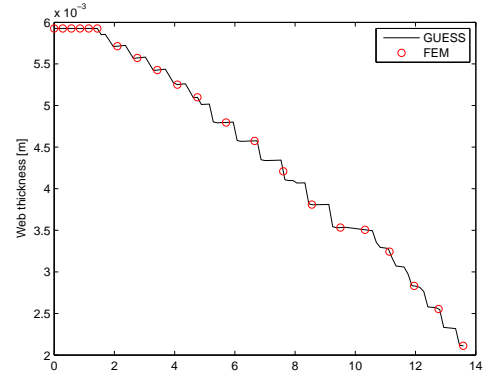
through an INCLUDE card.

A typical main file is:

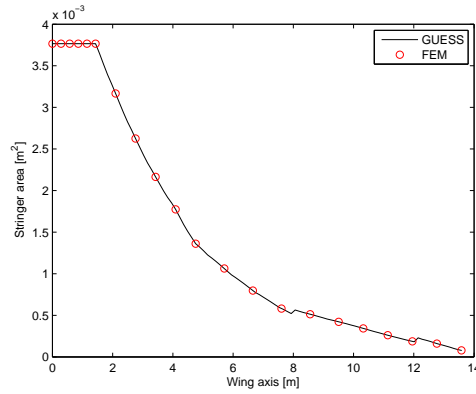
```
SOL 144
$ aero reference values
AEROS          0      2.2488  27.18   61      0      0
$ include maneuver file
INCLUDE man25_nmax.inc
```



(a) Skin



(b) Web



(c) Stringer

Figure A.10: Structural properties interpolation.

```
$ include configuration file
INCLUDE test_femCONM_CONF1.inc
```

This will avoid mixing loads and inertial configurations, making sure mass and maneuver loads provided to FEMGEN are consistent, i.e. the maneuver load takes into account the inertia relief of the mass configuration considered. The following section will consider the transfer of secondary masses. Sec. A.1.4 will show how to export external loads from SMARTCAD to FEMGEN.

Export masses

A dedicated function within FEMGEN package allows to transfer masses from SMARTCAD to FEMGEN. The function `load_ribs_CONM` takes as input SMARTCAD filename and the file `ribs_data.mat` saved by FEMGEN. This file has few information about ribs and their mesh discretization. If FEMGEN is run again with different settings for ribs, the file will be overwritten with the new values. Thus the user must make sure the `ribs_data.mat` file used is consistent with the FE model being considered.

Two files will be created:

- `ns_mass.dat`;

- `RBE3.dat`.

The former defines a list of CONM1 and GRID points, while the latter RBE3 elements linking the new grids to the vertexes of the FE ribs. The former file is referred by the latter through an INCLUDE card. The grid points added to the FE model are simply the same nodes of the SMARTCAD stick model. They have indeed the same coordinates as well as same ID and carry the CONM1 masses. The program connects each mass to the nearest rib.

Export loads

Successively, the user created a load set to apply by running one the following solvers within SMARTCAD :

- `solve_free_lin_trim` to run static trim for rigid and deformable aircraft. Use the main `.inc` file with masses and maneuvers included;
- `solve_free_lin_trim_guess_std` to run static trim for rigid aircraft only. Use the file `gstd_model.dat` with maneuvers included. This option consider the MTOW configuration as default. The aircraft in this case is represented by a single mass located at center of gravity. The user can of course modify the value of the mass and its position to consider other mass configurations.
- `solve_vlm_rigid` to run a simple aerodynamic condition in terms of incidence, sideslip and angular velocities. This solver simply provides loads for a prescribed flight condition and has no dependence upon mass configuration.

A dedicated function `neo2rbeforce_beam2` within NeoCASS package allows to collect forces along SMARTCAD aerodynamic mesh. Please see the dedicated help for instructions (type `help neo2rbeforce_beam2` at the prompt) about this function. Also, the file `README.txt` in FEMGEN directory reports some important comments. Basically, the program considers the results from SMARTCAD in terms of pressure forces along the aerodynamic mesh. Then, a beam-spline is used to connect the provided structural nodes to the aerodynamic points where singularities are located. The function exports the aerodynamic load as a lumped set of forces and moments (using FORCE and MOMENT cards). The final LOAD ID is defined as input-parameter.

In the example below, a LOAD ID equal to 1 has been required. The function combines LOAD 100, 101 and 102 into the final set 1. A dummy node is also created to define the axis of rotation for angular rates and accelerations. This point is the one defined by the SUPORT card in SMARTCAD file. Usually, the nodes created by the previous program `load_ribs_CONM` are used as load points for the FE model as well. Of course, the user can give as inputs the structural nodes of the whole FE model but this is not suggested because it may lead to unreal local displacements.

If either of the trim solver has been run, maneuver accelerations and angular velocities will be exported, so that NASTRAN can correctly include inertial loads. In this respect, GRAV and RFORCE cards are used. An excerpt of the output file is:

```
LOAD          1  1.0000  1.0000      100  1.0000      101  1.0000      102
$
```

```

$ Maneuver inertial loads
$
$-----2-----3-----4-----5-----6-----7-----8-----9-----10
GRID          2017          0 11.8172 11.8172 -0.4651          0 123456
GRAV          101          0 24.5250 -0.0000 -0.0000 -1.0000
RFORCE        102    2017          0 0.0000 1.0000 0.0000 0.0000
                -0.0000
$-----2-----3-----4-----5-----6-----7-----8-----9-----10
FORCE,100,2000,,11065.895063,0.000000,-0.029062,0.999578
MOMENT,100,2000,,4027.353586,0.388064,0.921243,0.026784

```

In this example, a pull up at $n_z = 2.5$ has been considered and loads at node 2000 only are reported. Neither angular acceleration nor angular rate has been given. The main file for a typical SOL 101 will be:

```

SOL      101
TIME     1000
CEND
LINE = 999999
SPCFORCES=ALL
ELFORCE=ALL
TITLE = MSC/NASTRAN : 3D wing box static analysis
ECHO=PUNCH(NEWBULK)
SPC = 1
LOAD = 1
ELSTRESS(CENTER)=ALL
DISPLACEMENT=ALL
BEGIN BULK
PARAM,GRDPNT,0
INCLUDE 'grid.dat'
INCLUDE 'element.dat'
INCLUDE 'beam.dat'
INCLUDE 'pshell.dat'
INCLUDE 'mat.dat'
INCLUDE 'rib.dat'
INCLUDE 'RBE3.dat'
INCLUDE 'spc_fuse.dat'
INCLUDE 'spc_sym.dat'
INCLUDE 'pbeam.dat'
INCLUDE 'pullup_load.dat'
ENDDATA

```

This file is created by FEMGEN. The user simply has to include the file generated by `load_ribs_CONM` (in this case named `pullup_load.dat`) and make sure the `LOAD=` card in the executive file section refer to the `LOAD` set available in `pullup_load.dat`. Of course the process can be repeated and several load conditions can be included through a `SUBCASE` statement:

```

SOL      101
TIME     1000
CEND
LINE = 999999
SPCFORCES=ALL
ELFORCE=ALL
TITLE = MSC/NASTRAN : 3D wing box static analysis
ECHO=PUNCH(NEWBULK)
SPC = 1
SUBCASE 1
LOAD = 1
SUBCASE 2
LOAD = 2
SUBCASE 3
LOAD = 3

```

The same happens for optimization runs:

```

SOL      200
TIME     1000
CEND
LINE = 999999
SPCFORCES=ALL
ELFORCE=ALL
TITLE = MSC/NASTRAN : 3D wing box optimization
ECHO=PUNCH(NEWBULK)
DESOBJ(MIN) = 1000000
ANALYSIS = STATICS
SUBCASE 1
SPC = 1
LOAD = 1
DESSUB = 1
ELSTRESS(CENTER)=ALL
DISPLACEMENT=ALL
BEGIN BULK
PARAM,GRDPNT,0
DSCREEN STRESS  -0.5    3
DSCREEN EQUA    -0.5    3
DOPTPRM DESMAX  50      DELP    5.0e-2  DPMIN  5.0e-4  DELX    2.5e-2
          DXMIN  2.5e-2  P1      1      P2      13      P2CALL 150
          APRCOD 1
INCLUDE 'grid.dat'
INCLUDE 'element.dat'
INCLUDE 'beam.dat'
INCLUDE 'pshell.dat'
INCLUDE 'mat.dat'
INCLUDE 'rib.dat'

```

```

INCLUDE 'RBE3.dat'
INCLUDE 'spc_fuse.dat'
INCLUDE 'spc_sym.dat'
INCLUDE 'desvar.dat'
INCLUDE 'dvprel1.dat'
INCLUDE 'dvprel2.dat'
INCLUDE 'dresp1.dat'
INCLUDE 'dconstr.dat'
INCLUDE 'analytical_buckling.dat'
INCLUDE 'pbeam.dat'
INCLUDE 'dtable.dat'
INCLUDE 'pullup_load.dat'
ENDDATA

```

Again, the main file is created by FEMGEN. The user can add other SUBCASES and LOAD cards for multiple cases.

A.2 Testcase

In this section, few comparisons between NeoCASS and FEMGEN models are reported. The purpose is to assess the data are correctly transferred from the stick model to the FE model. Fig. A.11 shows the stick model for the wing considered. It has been extracted from the stick model of the full aircraft. Constraints applied are:

- symmetry constraints at midplane (T2, T4, T6);
- horizontal and vertical displacement, pitch rotation at wing-fuselage intersection (T1, T3, T5).

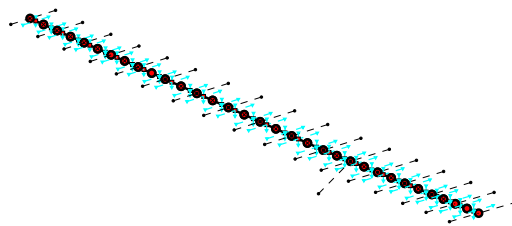


Figure A.11: SMARTCAD structural model of the wing.

The first check is for inertias. Table A.1 compares the total value of mass, the position of the center of gravity (CG) and the main moments of inertia for the wing-box only. The agreement is fairly good, considering the structural properties from GUESS are interpolated along the bays of the wing-box which results in a coarser representation of the trend

(see Fig. A.10). Also, the stick model considers a rectangular section while the FE has a more accurate description of the skin. Table A.2 shows the same results for the wing-box with secondary and non-structural masses added. The second test considers wing-box

	Mass [Kg]	X_{CG} [m]	Y_{CG} [m]	Z_{CG} [m]	I_{XX} [m ⁴]	I_{YY} [m ⁴]	I_{ZZ} [m ⁴]
SMA	1376.988	11.801	3.961	1.485	1.416e+4	0.015e+4	1.415e+4
NAS	1558.218	11.817	3.933	1.573	1.559e+4	0.021e+4	1.573e+4

Table A.1: Inertial properties for wing-box only.

	Mass [Kg]	X_{CG} [m]	Y_{CG} [m]	Z_{CG} [m]	I_{XX} [m ⁴]	I_{YY} [m ⁴]	I_{ZZ} [m ⁴]
SMA	5948.410	11.688	4.687	1.342	4.088e+4	0.403e+4	4.280e+4
NAS	5767.179	11.696	4.658	1.347	4.222e+4	0.217e+4	4.327e+4

Table A.2: Inertial properties for wing-box and lumped masses.

vibration modes, comparing the results from NASTRAN SOL 103 and the `solve_eig` solver in SMARTCAD . Considering the approximation introduced above, the results are satisfactory. Table A.3 reports the frequencies of the first four modes. Modal shapes determine by SMARTCAD are depicted in Fig. A.12. Modes depend on the inertia and

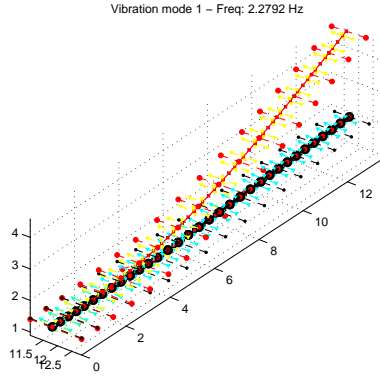
	F1 [Hz]	F2 [Hz]	F3 [Hz]	F4 [Hz]
SMA	2.279	5.341	6.961	11.770
NAS	2.267	6.149	7.722	16.706

Table A.3: Comparison for vibration modes.

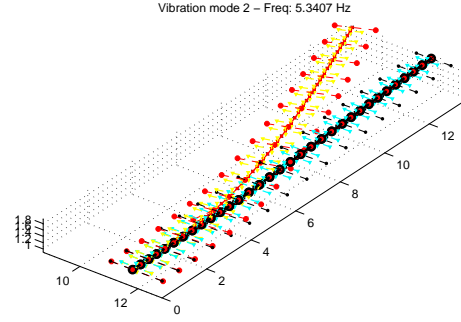
stiffness distribution, i.e. they are proportional to generalized stiffness and to the inverse of the generalized mass. The results seem to imply the FE model has more stiffness than the stick model. Finally, Fig. A.13 shows the displacements for the nodes of the stick model. In the same figures, the same points belonging to the FE model and connected to the ribs are presented. Again, the results have a fairly good agreement. To conclude, an example of static load response is presented. In this case, a pull-up maneuver has been solved considering the full stick model of the flying aircraft. The deformed shape at trim condition is given in Fig. A.14. Loads are then exported through the dedicated function (see Sec. A.1.4) and a static solution SOL 101 in NASTRAN carried out. Again, the displacements of the stick nodes connected to the FE ribs are reported. As mentioned above, the FE model features more stiffness than the stick model, thus leading to lower displacements when the tip is approached.

A.3 Conclusions

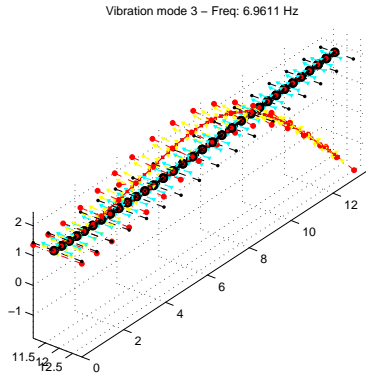
The present report has outlined the main features of FEMGEN, Finite Element GENerator. The program is placed downstream of NeoCASS procedures and raises the fidelity of



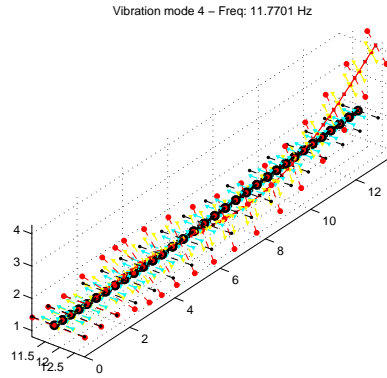
(a) Mode 1: first bending



(b) Mode 2: inplane bending



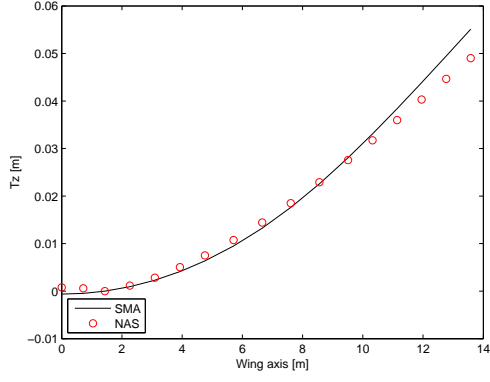
(c) Mode 3: second bending



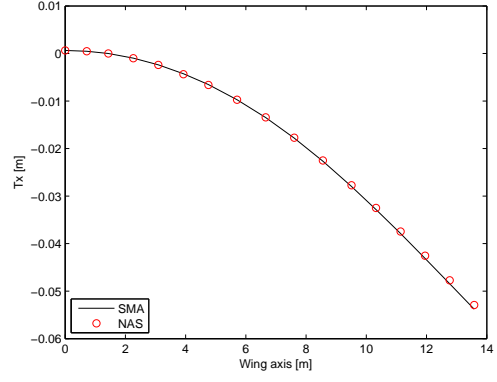
(d) Mode 4: first torsional

Figure A.12: Vibration modes computed in SMARTCAD (magnified by 50).

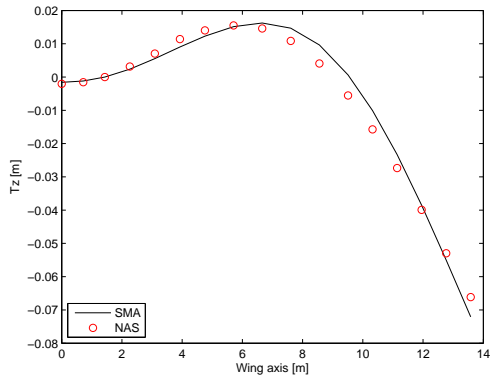
numerical analysis of the wing-box using through a complete FE model. FEMGEN allows different kind of analyses by means of NASTRAN: static, flutter, vibration modes and optimization. Three steps are required. At first FEMGEN is run and the FE model is created. The geometry of the wing-box is loaded from the `.xml` file. Structural properties are initialized from the stick model available in GUESS. Then external non-structural masses are included in the model and different set of loads from SMARTCAD can be defined. The result is a model suitable for accurate static and dynamic analyses.



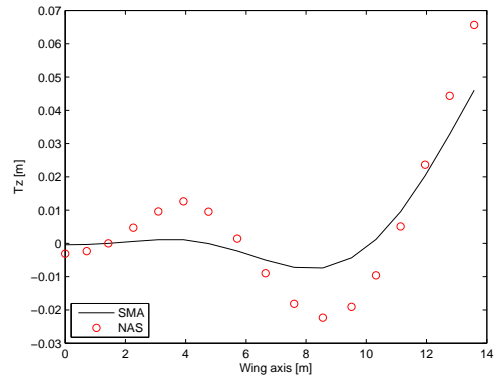
(a) Mode 1



(b) Mode 2



(c) Mode 3



(d) Mode 4

Figure A.13: Vibration modes.

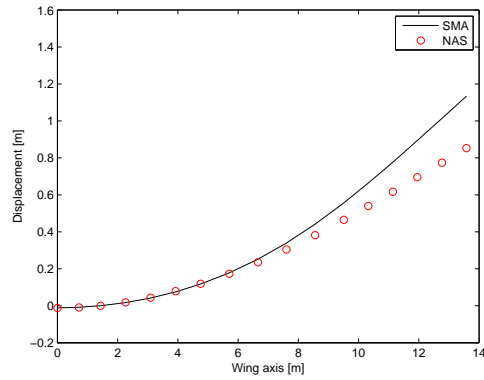


Figure A.14: Deformed shape for static pull-up, $n_z = 2.5$.

Bibliography

- [1] M.D. Ardema, A.P.P. Chambers, A.S. Hahn, H. Miura, and M.D. Moore, *Analytical Fuselage and Wing Weight Estimation of Transport Aircraft*, Tech. Report 110392, NASA, Ames Research Center, Moffett Field, California, May 1996.
- [2] G. Bindolino, G. Ghiringhelli, S. Ricci, and M. Terraneo, *Multilevel structural optimization for preliminary wing-box estimation*, Journal of Aircraft **47** (2010), no. 2, 475–489.
- [3] L. Cavagna, S. Ricci, and L. Riccobene, *A fast tool for structural sizing, aeroelastic analysis and optimization in aircraft conceptual design*, AIAA-2009-2571 (Palm Springs, California, USA), 50th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials (SDM) Conference, 4 – 7 May 2009.
- [4] MSC, *Msc/nastran encyclopedia v 70.5*, May 1998.