NeoCASS Overview

Version 2.0.125

November 2011
Outline

1. NeoCASS overview                                 pag. 3
2. AcBuilder overview                                pag. 25
3. NeoCASS GUI overview                              pag. 45
NeoCASS overview

*NeoCASS* (Next generation Conceptual Aero-Structural Sizing Suite) is a collection of Matlab® analysis modules for:

- Initial aircraft structural sizing;
- Modal analysis;
- Linear/non-linear static analysis;
- Aeroelastic analysis (static aeroelasticity, flutter);

Connected with tools for:

- Spatial coupling (MLS and RBF);
- Aerodynamic analysis (internal VLM/DLM);
- MDO.

Interfaced to:

- External codes (Edge-FOI, MSC/NASTRAN, others);
- Other modules of Ceasiom.
NeoCASS overview

Step by step NeoCASS sequence of operations:

1. Input of **Aircraft Geometric description** and **technological solutions** from AcBuilder module through XML file;
2. Input of **Sizing Mode**;
3. Initial structural sizing;
4. Structural Analysis;
5. Aeroelastic analysis, including MDO;
6. Output: **vibration modes, trimmed elastic aircraft, aeroelastic derivatives, flutter boundaries, divergence speed, aileron reversal, corrected inertia properties**
AcBuilder module

**AcBuilder** (Aircraft Builder) is:

- A graphical editor to prepare the aircraft XML file requested by NeoCASS and other modules of Ceasiom.
- The standard pre-processor for **NeoCASS**.
Different layout available
GUESS: a module for the initial sizing

GUESS (Generic Unknowns Estimator in Structural Sizing)

Why the need for a ‘GUESS’ initial sizing?
- Since the weight estimating formulas are based on existing aircraft, their application to unconventional configuration (i.e., canard aircraft) is difficult;
- The impact of advanced technologies and materials cannot be assessed in a straightforward way.

GUESS enhances the NASA-PDCYL code by Ardema [1] with:
- Sizing of tail surfaces (FAR-25);
- Calculation of torsional and shear structural properties through semi-monocoque theory;
- Non-conventional aircraft.

GUESS : a module for the initial sizing (contd)

The total amount of load-bearing structural weight is determined on:

- Real material properties
- Real aircraft layout (conventional or not)
- Real load conditions

Two methods implemented:
- **Standard mode**: predefined sizing maneuvers, simplified aerodynamics;
- **Modified mode**: user-defined maneuvers, rigid aircraft trimming and VLM-based aerodynamics.

Once loads are determined on rigid aircraft (using simplified or VLM-based aerodynamics), **GUESS** sizes each component section by section.

Limits:
- No aeroelastic effects considered;
- Fully-stressed sizing based on max. stress and/or instability loads (local-global) and isotropic materials.
Structural layout adopted by **GUESS**:  
- Multi-web structure for lifting surfaces and carrythrough component;  
- Stiffened framed-unframed shell for fuselage;  
- **NEW!** A typical wingbox section (available since version 2.0 of **NEOCASS**)

Once computed the structural weight, the total weight is obtained by summing up the non structural weights resulting from **W&B** module.
Lifting surfaces sizing

Features:
- The minimum amount of material is determined for each section
- **GUESS** adopts different concepts of multi-web box beam with span-wise webs
- Each concept is analytically proportioned to maximize different performances (buckling, minimum gage,...)
- A regression takes into account secondary structural weight (joints, fasteners, landing gear support, bulkheads, high lift devices...)

<table>
<thead>
<tr>
<th>$K_{con}$</th>
<th>Covers</th>
<th>Webs</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Unstiffened</td>
<td>Truss</td>
</tr>
<tr>
<td>2</td>
<td>Unstiffened</td>
<td>Unflanged</td>
</tr>
<tr>
<td>3</td>
<td>Unstiffened</td>
<td>Z-stiffened</td>
</tr>
<tr>
<td>4</td>
<td>Truss</td>
<td>Truss</td>
</tr>
<tr>
<td>5</td>
<td>Truss</td>
<td>Unflanged</td>
</tr>
<tr>
<td>6</td>
<td>Truss</td>
<td>Z-stiffened</td>
</tr>
</tbody>
</table>

Sketch of the wing concept

Lifting surfaces structural concepts

NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 11
Fuselage sizing

Features:
- **GUESS** adopts different concepts of stiffened shells with/without frames
- A regression takes into account secondary structural weight (flooring, pressure webs, ...)

<table>
<thead>
<tr>
<th>$K_{con}$</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Simply stiffened shell, frames sized for minimum weight in buckling</td>
</tr>
<tr>
<td>2</td>
<td>Z-stiffened shell, frames best buckling</td>
</tr>
<tr>
<td>3</td>
<td>Z-stiffened shell, frames buckling-minimum gage compromise</td>
</tr>
<tr>
<td>4</td>
<td>Z-stiffened shell, frames buckling-pressure compromise</td>
</tr>
<tr>
<td>5</td>
<td>Truss-core sandwich, frames best buckling</td>
</tr>
<tr>
<td>6</td>
<td>Truss-core sandwich, no frames</td>
</tr>
<tr>
<td>7</td>
<td>Truss-core sandwich, no frames, buckling-minimum gage-pressure compromise</td>
</tr>
</tbody>
</table>

Fuselage structural concepts
The NEW wingbox structural layout for lifting surfaces

- Implements semi-monocoque concept.
- Three main design variable adopted during initial sizing with GUESS.
- Possible generalization up to 10 design variables during MDO.
- Strength and stability constraints taken into account during initial sizing.

New Wingbox Section Model (kcon=9)

\[ A, I_{\text{cap_front_upp}}, A, I_{\text{str_upp}}, A, I_{\text{cap_rear_upp}}, t_{\text{web_front}}, t_{\text{skin_upp}}, t_{\text{web_rear}}, t_{\text{skin_low}}, A, I_{\text{cap_front_low}}, A, I_{\text{str_low}}, A, I_{\text{cap_rear_low}} \]
The NEW wingbox structural layout for lifting surfaces

- Implements semi-monocoque concept.
- Three main design variable adopted during initial sizing with Guess.
- Possible generalization up to 10 design variables during MDO.
- Strength and stability constraints taken into account during initial sizing.

New Wingbox Section Model (kcon=9)

- $T_{web}$ (Shear)
- $T_{skin}$ (Torque)
- $A$ (Bending)
The NEW wingbox structural layout for lifting surfaces

- Two optional parameters can be included into XML file to specify (for each lifting surface) the number of stiffeners and the ribs pitch.
  
  \[
  \text{user\_input.material\_property.wing.nstr} \\
  \text{user\_input.material\_property.wing.rpitch}
  \]

- If not included, the code automatically defines ribs pitch = 0.55 m and stiffeners pitch comprised between .16 and .17 m.

- **IMPORTANT**: to select option kcon=9 the Matlab Optimization Toolbox must be installed on the computer running NeoCASS.
GUESS: automatic stick model generation

Items automatically generated:
- Stick model and mechanical properties;
- Stress-recovery points;
- Extra-nodes for Fluid-Structure coupling.

Stick model based on a non-linear, three nodes, finite volume beam model.
SMARTCAD overview

**SMARTCAD** (*S*implified *M*odels for *A*eroelasticity in *C*onceptual *A*ircraft *D*esign)

- **Input:** ASCII files derived from NASTRAN® formats. Why?
  - Platform independent;
  - To avoid wasting time to define and learn a new format;
  - Commercial pre/post-processors can be used to visualize the model and results;
  - **SMARTCAD** can be almost easily bypassed in favor of NASTRAN® without precluding the overall functionality of CEASIOM design tool;
  - The comparison with the validated commercial code is then straightforward.
SMARTCAD overview

Once available, the aeroelastic model can be processed by SMARTCAD to compute:

- Static aeroelasticity;
  - Divergence speed;
  - Deformable trimmed configuration;
  - Flexible stability derivatives.

- Dynamic aeroelasticity;
  - Flutter diagram (V-g plot);
  - Flutter envelope.

- MDO, to improve any of the aeroelastic responses by changing the structural properties initially estimated by GUESS.
SMARTCAD: static aeroelastic problem formulation

- NASTRAN-like formulation (mean axes formulation, to nullify the inertial coupling and a quasi-steady approximation for aerodynamic forces);
- Control surface mechanical stiffness and damping are neglected;
- Six reference conditions have to be defined in order to correctly couple the reference motion and the structural one;
- In case of multi-control surfaces linear constraints are introduced to reduce the total number of degrees of freedom for trimming;
- Typical results: deformable/rigid stability derivatives, free flying aircraft trimmed configuration, divergence speed.
Pull-up maneuver
Trim results, $M_\infty=0.5$, $n_z=3.1$, sea level

<table>
<thead>
<tr>
<th>Rigid [deg]</th>
<th>Deformable [deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>$\theta_{\text{stab}}$</td>
</tr>
<tr>
<td>11.0</td>
<td>-10.0</td>
</tr>
</tbody>
</table>
SMARTCAD: linearized flutter analysis

- Typical assumptions based on assumed structural shapes, the availability of an aerodynamic transfer matrix $H_{am}$ and the Laplace domain $s$; static corrections available (T-Tail).

- The flutter system of equations is solved in a continuative way, i.e. as a nonlinear algebraic system of equations with the eigenvalue and the eigenvector as unknowns.

- Particularly suitable for optimization purposes since it unifies analysis and sensitivity calculation in a very effective way.

- Eigenderivatives come from the solution of a linear system of equations determined by differentiating the nonlinear equations with respect to a structural parameters.

- The coefficient matrix to be used is the same as the one used for the flutter-tracking process but with a different right hand side.
SMARTCAD: linearized flutter analysis

Clamped node

Model used for TCR tail

Fin bending mode 1: 1.6 Hz

Hor. Tail “roll” mode 3: 4.3 Hz

<table>
<thead>
<tr>
<th>Mode</th>
<th>SMARTCAD</th>
<th>NAISTRAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.60</td>
<td>1.60</td>
</tr>
<tr>
<td>2</td>
<td>2.63</td>
<td>2.62</td>
</tr>
<tr>
<td>3</td>
<td>4.32</td>
<td>4.34</td>
</tr>
<tr>
<td>4</td>
<td>4.63</td>
<td>4.59</td>
</tr>
<tr>
<td>5</td>
<td>8.16</td>
<td>8.16</td>
</tr>
<tr>
<td>6</td>
<td>8.71</td>
<td>8.69</td>
</tr>
<tr>
<td>7</td>
<td>13.32</td>
<td>13.25</td>
</tr>
<tr>
<td>8</td>
<td>18.87</td>
<td>18.10</td>
</tr>
<tr>
<td>9</td>
<td>18.93</td>
<td>18.76</td>
</tr>
<tr>
<td>10</td>
<td>20.07</td>
<td>21.48</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mach</th>
<th>SMARTCAD</th>
<th>NAISTRAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.50</td>
<td>5.66·10⁴</td>
<td>6.55·10⁴</td>
</tr>
<tr>
<td>0.70</td>
<td>5.54·10⁴</td>
<td>6.43·10⁴</td>
</tr>
<tr>
<td>0.85</td>
<td>5.43·10⁴</td>
<td>6.16·10⁴</td>
</tr>
<tr>
<td>0.97</td>
<td>5.38·10⁴</td>
<td>5.92·10⁴</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mach</th>
<th>SMARTCAD</th>
<th>NAISTRAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.50</td>
<td>3.46</td>
<td>3.61</td>
</tr>
<tr>
<td>0.70</td>
<td>3.43</td>
<td>3.56</td>
</tr>
<tr>
<td>0.85</td>
<td>3.38</td>
<td>3.49</td>
</tr>
<tr>
<td>0.97</td>
<td>3.25</td>
<td>3.39</td>
</tr>
</tbody>
</table>

Flutter dynamic pressure [Pa]

Flutter frequency [Hz]
References


AcBuilder Overview
AcBuilder: overview

What is AcBuilder?
- AcBuilder is a graphic editor of XML file including some CAD capabilities.
- It is a tool running under Matlab used to support the user in the preparation of aircraft XML file to be processed by CEASIOM or its single modules like NeoCASS.
- It is the standard pre-processor for NeoCASS module.

What are the requested inputs to run AcBuilder?
- The geometric data describing the aircraft.

What are the typical outputs produced by AcBuilder?
- A single .XML file that describes completely the aircraft.
- Snapshots of different windows.
**AcBuilder: Menus structure**

AcBuilder is driven by a top level **Main Menu**, some contextual **Side Menus** and a table-like **Panel** to insert the numeric data.

The top level Main Menu is composed by the following items:

- **Project**
  - Set default/Import XML/Export XML/Take screenshot/Save data and close/Close window

- **View**
  - Reset/Side/Front/Top/Background color

- **Geometry**
  - Component/Fuel/Geometry output/Export SDSA geometry

- **Weight & balance**
  - Weight & Balance/Center of gravity

- **Technology**
  - Technology/Import XML
**AcBuilder: Menus goals**

- **Project**
  - To open, import, export and save the aircraft XML file and to take some screenshots

- **View**
  - To select the view style and to change the Background color

- **Geometry**
  - To define the geometry of each aircraft’s component; to define the size of fuel tanks and wingbox; to compute all the relevant geometry properties

- **Weight & balance**
  - To define the cabin properties; to define the known masses; to compute the weight & balance properties

- **Technology (used only for NeoCASS)**
  - To define the structural and aerodynamic meshes; to define the material properties; to define some sizing conditions.
**AcBuilder: tips & tricks**

- When Acbuilder is invoked from the Matlab command window by entering AcBuilder, a *template* aircraft is automatically loaded. In this way, a set of reference data is already defined. The user has simply to modify the data, checking on the screen the results of his changes.
- The default values for Template aircraft are defined in the Matlab routine named *acb_initac.m*
- The airfoil data are included in a subdirectory named airfoil. The full path of this directory must be defined into the matlab routine *acbuilder.m* (variable *afpath*)
- Press Left (or right) mouse button to rotate the aircraft.
- Press CTRL Left (or CTRL right) mouse button to move the aircraft.
**AcBuilder**: menus and windows

- **Main Menu window**
- **Graphic window**
- **Aircraft Components window**
- **Geometry data window**

NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 29
AcBuilder: menus and windows

Main Menu: Project
**AcBuilder:** menus and windows

**Main Menu: View**

[Image of a 3D model of an aircraft with a menu interface showing various components and parameters.]
AcBuilder: menus and windows

Main Menu: Geometry/Component
AcBuilder: menus and windows

Main Menu: Geometry/Fuel

Fuel Menu: Fuel Tanks definitions
**AcBuilder: menus and windows**

**Main Menu:** **Geometry/Fuel**

**Fuel Menu:** **Wingbox definitions**

Image of a 3D model of an aircraft with highlighted fuel tanks and parameters in the wingbox definitions window.
AcBuilder: menus and windows

Main Menu: Weight & Balance
AcBuilder: menus and windows

Main Menu: Weight & Balance/Weight & Balance

Weight & balance Menu: Mandatory parameters
AcBuilder: menus and windows

Main Menu: Weight & Balance

Weight & balance Menu: Miscellaneous
AcBuilder: menus and windows

Main Menu: **Weight & Balance/ Centers of Gravity**

Centers of gravity Menu: List of items

Values Menu: Coord. of CoG
**AcBuilder:** menus and windows

**Main Menu:** Technology

**Technology Menu:** Geometry (beam_model)

**Parameters Menu:** Beam mesh definition
AcBuilder: menus and windows

Main Menu: Technology

Technology Menu: Geometry (aero_model)

Parameters Menu: VLM/DLM mesh definition
**AcBuilder:** menus and windows

**Main Menu:** Technology

**Technology Menu:** Geometry (spar_location)

**Parameters Menu:** Wingbox definition for all items

---

*NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 42*
AcBuilder: menus and windows

Main Menu: Technology

Technology Menu: Material

Parameters Menu: Material properties for all items
NeoCASS GUI Overview

Version 2.0.125

November 2011
GUI interface to NeoCASS Suite is based on four main Panels, i.e.:

- File;
- Settings;
- Run;
- Results;

and different sub-panels and windows to help the user in the correct definition of input data. Using these user-friendly menu, the user could introduce all parameters requested by different analysis modules, while the order of the GUI panels well reproduces the typical analysis sequence.
NeoCASS GUI – FILE Panel

- The FILE Panel is the first one that appears when NeoCASS code is invoked. By means of this panel it is possible to perform the following actions:
  - Read input files requested by GUESS, run GUESS code and generate the aircraft stick model.
  - Add Reference Values for geometrical and aerodynamic parameters.
  - Select type of analysis to be run and related input/output data.
  - Open a previously saved NeoCASS data base (Matlab format).
  - Save all input data into a new SMARTCAD input file (ascii .dat file).
  - Open an already available SMARTCAD input file (ascii .dat file).
  - Select among enabled solvers which one must be actually executed.
NeoCASS GUI – FILE Panel

Button for opening an already existent NeoCASS project

Definition of setting parameters for different type of analyses

Assembly a SMARTCAD analysis file

GUESS Subpanel: Files input, Edit and RUN GUESS

Open and Edit SMARTCAD files

List of enabled solvers
Open aircraft window

After pressing the **Open aircraft button** a select file window appears where the user can select the .XML file describing the aircraft to be designed.
Sizing mode window

After pressing the **Sizing mode button** a new window appears, enabling the user to select among two options: run **GUESS** in **Standard mode** (using predefined maneuver for structural sizing) or in **Modify mode**, allowing for the possibility to define different maneuvers to be used by **GUESS** during the first structural sizing.

File where the maneuvers are saved

---

NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 49
After pressing the **RUN GUESS button** a select file window appears where the user has to provide the file name where the aircraft geometry is saved after sizing (.inc ascii file).
Before running **GUESS**, the system shows a new window, named ChEcK, with different options for the user.

- **Models visualization options**
  - Horizontal tail all movable
  - Vertical tail all movable
  - Canard all movable

- **Definition of all movable control surfaces**

- **Definition of different mass configurations**
By pressing the **Aerodynamic model button** it is possible to visualize the aerodynamic mesh including the control surfaces.
By pressing the **Structural model button** it is possible to visualize the internal (analytical) and exported (stick model) structural models.
Mass configuration window

Using the **Mass configuration button** it is possible to define different mass configurations, in terms of Fuel and Passengers. Only in the case of **GUESS** in Modify mode it is possible to link each mass configuration to a specific sizing maneuver.

Linking between Mass configuration and Load condition (Sizing maneuvers).
The Subpanel REFERENCE Settings must be used to input reference parameters used for the aerodynamic calculations, i.e.:

- Reference **Chord** (CREF);
- Reference **Span** (BREF);
- Reference **Surface** (SREF);
- Vertical Symmetry (0 Full model, 1 half model);
- Horizontal Symmetry (0 No Ground, 1 Ground effect);
- Height (active if Ground Effect is selected);
- Kernel order for DLM solver (1 Linear, 2 Quadratic).
Reference and Aerodynamic Settings Subpanel

Reference values settings

Aerodynamic settings

NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 56
The GUI Subpanel ANALYSIS Settings must be used to select which kind of analysis must be run and to input the requested parameters. The GUI Subpanel could be divided into three small panels, related to the following type of analysis:

- Static aeroelastic analysis;
- Modal analysis;
- Flutter analysis;
Analysis Settings Subpanel

- Input data for static aeroelastic analysis
- Input data for modal analysis
- Input data for flutter analysis
Analysis Settings Subpanel: Modal Analysis

The parameters that must be provided to run a Modal Analysis are the following:

- **Normalization (1 MASS, 2 MAX, 3 POINT):** in case a POINT normalization is chosen the user must provide the Grid Point ID and DOF with respect to the normalization is done;
- **ID:** Grid Identification Number;
- **DOF (1,2,3,4,5,6);**
- **LMODES:** Number of modes retained during modal calculations;
- **From - To:** an alternative way to define the bandwidth of interest (lower and upper frequencies).
Analysis Settings Subpanel: Modal Analysis

Normalization menu

Grid ID and DOF, in case of POINT normalization

Number of modes

Frequency bandwidth

NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 60
Analysis Settings Subpanel: Static Aeroelasticity

When static Aeroelastic Analysis is selected, three are the requested input parameters:

- Selection between typical (symmetric or asymmetric) or custom maneuvers, by pressing the related checkboxes. Using the first option only the minimum set of parameters necessary to solve the trim problem is requested as input. Using the second option, all the fields included into the TRIM card are requested;
- Number of Flight Conditions;
- Values for the TRIM card parameters for each flight condition.
Analysis Settings Subpanel: Static Aeroelasticity

Radio button to select Static Aeroelastic Analysis

Number of flight conditions

In this field the SUPORT GRID has to be specified. The geometry file generated by GUESS, containing the stick model, reports in the last rows the suggested GRID to be used as SUPORT.
In case of typical maneuvers have been selected, the following window is issued as many times as the number of flight conditions input by the users. For each kind of maneuver only the parameters necessary to solve the trim problem are requested.

Flight and control states necessary to define the prescribed maneuver.
When custom maneuvers option is selected, the complete table of all flight states must be filled by the user in a consistent way, so to be able to solve the trim problem that in general is stated as a system of 6 equilibrium equations for a free flying aircraft.
Analysis Settings Subpanel: Flutter Analysis

When Flutter Analysis is selected, the user first of all must specify the number of reduced frequencies (max. 12) and insert their values in the Table that appears after pressing the button INSERT Values. The minimum reduced frequency is automatically set to 0.001 but it can be modified by the user. Then, the user must specify the number and list of modes to be retained for the calculation of Generalized Aerodynamic Forces matrix (Qhh) and the number and list of modes to be tracked during the V-g plot calculation. Finally, users must choose between two possibilities: Flutter analysis for a single assigned flight condition or Flutter Envelope for an assigned number of Mach values. In the first case (generation of V-g plot) the requested input parameters are:

- Max Speed for flutter calculation;
- Max V step (number of steps used during iterative mode tracking);
- Air Density;
- Mach Number.
Analysis Settings Subpanel: Flutter Analysis

When Flutter Envelope is selected, the requested input parameters are:

- Number of Mach values for which flutter envelope is computed;
- Values of Mach numbers: when the Insert Values button is pressed a table appears where the user must insert Mach number values.
Analysis Settings Subpanel: Flutter Analysis

Radio buttons to select single flutter analysis (V-g plot) or flutter envelope

Number of reduced frequencies
Press to insert values

Selection of modes for calculation and V-g plot

Number of Mach numbers
Press to insert values

Analysis parameters for mode tracking

Flight data
When the **MSELECT Button** is pressed, a listbox window pops up showing the list of eigenvalues available (as set by LMODES card). The user can select the list of retained modes simply selecting different rows by pressing at the same time SHIFT or CTRL buttons.
Analysis Settings Subpanel: Flutter Analysis

When the **FMODES Button** is pressed, a listbox window pops up showing the list of eigenvalues available (as set by LMODES card). The user can select the list of modes he want to track during the $V-g$ plot calculation. For example, rigid modes can be retained into the modal basis during calculation of generalized forces but they cannot be tracked during the $V-g$ plot calculation.
Analysis Settings Subpanel: Flutter Analysis

Input reduced frequencies

Input Mach values

NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 70
Assembly a SMARTCAD analysis file

When the **ASSEMBLY Button** is pressed, a listbox window pops up showing all the .inc and .dat files available in the current directory. In this way it is possible to assemble a final SMARTCAD analysis file, simply by merging a stick model file with specific .inc files including the requested analysis cards. The assembly is done by means of the INCLUDE card that is automatically written on the final .dat file. Multi selection is possible using SHIFT and CTRL buttons.
Saving the .inc analysis files

After the input of all parameters for each kind of analysis, the user will be asked to save them into a .inc file. In this way it is possible to save different .inc files including different kind of analysis that later can be merged with a stick model file in a final .dat smartcad using the INCLUDE card.
General Settings Panel

The Panel Settings is used to input parameters for analysis solvers. In particular, the following parameters must be selected by the user:

- Structural Model (1 Linear Beam, 2 Equivalent Plate, 3 Non-Linear Beam). In the version 2.0 of NeoCASS the Equivalent Plate structural model is not active even if already included;
- Aspect Ratio: when Equivalent Plate is selected the user can control the size (and number) of Plate elements automatically generated by means of this parameter, which control the aspect ratio of Plate. It is set by default equal to 1;
- Sub-Iter: when Non-Linear beam is selected, Sub-Iter defines the number steps needed to reach convergence with an assigned load;
- contd…
General Settings Panel

- Conv. Tolerance (convergence error on the residue during non-linear analysis);
- Load Steps: number of load steps during static non-linear analysis or maximum number of coupled iterations during Static Aeroelastic Analysis;
- Under-relaxation (relaxation factor adopted transferring loads from aerodynamic to structural mesh): 0.5 is the default value.
General Settings Panel

Structural model selection

Non-linear analysis control parameters

Control of equivalent plate Aspect Ratio

Structural-aerodynamic coupling control parameter

NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 75
RUN Panel

The RUN Panel simply collects all buttons related to each solvers. Only buttons related to solvers for which all the requested data have been correctly input are active (clickable). In the same panel is located the button named Start used to start all solvers in the automatic analysis mode, when selected (option inactive for Version 2.0). The following analyses can be started by the panel:

- **STATIC**: Static analysis under aerodynamic and inertial relief loads;
- **MODAL**: Eigenvalues analysis;
- **TRIM**: Static aeroelasticity analysis: trim and aeroelastic stability derivatives;
- **FLUTTER**: Flutter analysis;
- **STEADY VLM/DLM**: Aerodynamic loads over rigid aircraft.
RUN Panel

Buttons to start available analyses

Button to start a sequence of analyses in automatic mode (inactive for the current version)
RESULTS Panel

RESULTS Panel is a collection of buttons and checkbox options allowing the user to analyze and post-processing the results of a NeoCASS run. Many of the buttons and selection fields available on this GUI Panel have a different meaning, depending on which kind of analysis has been performed. The post-processing options are the following:

- **GUESS**: By pressing the button **GUESS** it is possible to plot the results of a **GUESS** analysis. The selection of which kind of diagram has to be plotted is done by filling the Selected Set field, ranging in this case from 1 to 10;

- **Aerodynamic Matrix**: In case of Flutter analysis, by pressing the button Plot Aero Matrix it is possible to plot the component of Aerodynamic Generalized Forces (Qhh): in this case the user must supply the ROW and COL indices. The Selected Set field in this case allows the user chose among the different Mach numbers for which Qhh has been computed (Flutter Envelope option);
RESULTS Panel

- **Plot Model**: When a simple structural analysis has been performed (Modal Analysis), by pressing the button Plot Model a new figure showing the structural model is created. Otherwise, in case of a Steady Rigid Aerodynamic Analysis (VLM) the same button allows to see both structural and aerodynamic panels;

- **Plot Deformed Model**: In case of a simple structural analysis (Modal Analysis), pressing the button Plot Deformed Model it is possible to visualize the mode shapes. The number of mode to be plotted is as usual controlled by the Selected Set field, while the Scale factor field determines the amplitude of the deformed shape. It is possible to generate an animation for each mode shape, choosing the number of mode and the number of frames. Pressing button Export Mode Animation an .AVI file is created containing the vibration mode animation.
RESULTS Panel

- **Plot Flutter Diagrams**: In case of Flutter Analysis pressing the button Plot Flutter diagrams the figures reporting V-g plot and Flutter envelope are created (if related output has been requested);

- **Selection Checkboxes**: Three selection checkboxes are available, all related to the plot of aerodynamic panels. They allow to include or exclude into the plot the wake elements, the panels normals and the contour visualization.

- **SAVE NeoCASS Project**: Pressing this button all intermediate results and data, organized into separated MATLAB c structures, are saved into an unique MATLAB c binary file (.MAT). In this way, it is possible in any moment to read it by pressing the related button into the GUI Panel File so recovering all the available data.

- **Close ALL**: To Exit from NeoCASS and delete all temporary files.
RESULTS Panel

Plot **GUESS** results

Plot options for deformed model (scale factor), mode animation (total frames) and output selection (set number)

Save NeoCASS project

Clear all windows and data

Qhh matrix plot options

Selection checkboxes to include different items into the plot

NeoCASS Tutorial – V.2.0.125 – Rel.1 November 2011 - pag. 81