FEM export manual

Main window

The main window of the FEMexport software, for data export and FEM analysis case definition from Panukl program to CalculiX is shown on Fig. 1.

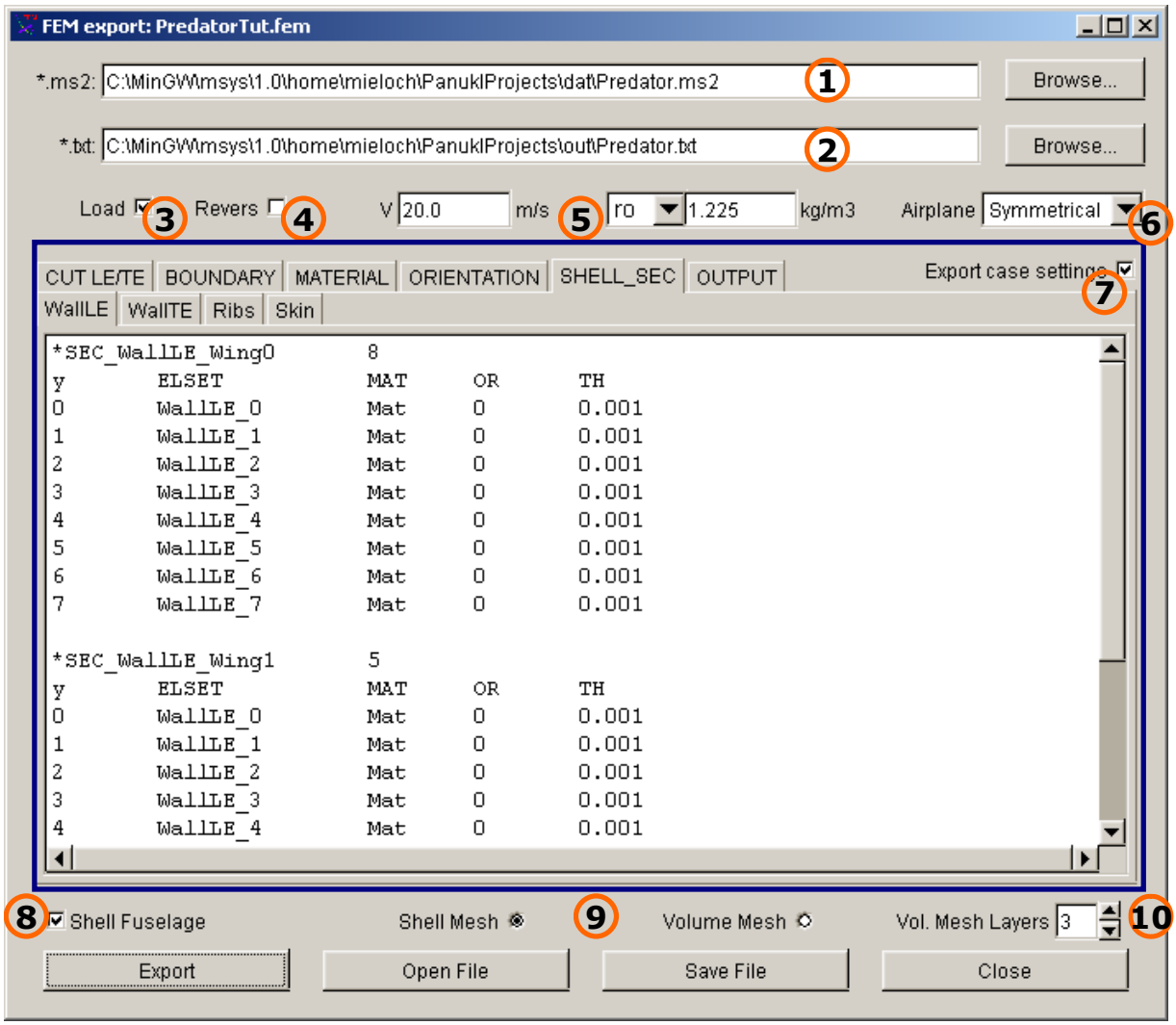

Fig. 1 Main window of the FEMexport software.

In the upper part of the window, path to .ms2 file with geometry is set ①. In ② path to file with data containing pressure coefficient distribution on panels .txt is set. Pressure distribution may be one of the loads in FEM analysis, but it's export can be turned off by "Load" flag ③. In CalculiX software normal vector of the panels is pointing from the airplane to the free space. Pressures in Panukl apply to convention in aerospace sciences, pressure with minus sign sucks the surface. In CalculiX the pressure sign has to be inverted for proper analysis. This is done automatically during export, but for visualization,

comparisons and verification purposes it can be inverted again with “Revers” flag ④. Nondimensional pressure coefficients from Panukl has to recalculated for pressure with units (1). This is done by setting speed of flight and air density. Air density can be determined explicit or from the flight level above ground ⑤. From the list ⑥ user can choose to export whole or only half of the airplane for FEM analysis.

$$p = Cp \cdot \frac{\rho \cdot V^2}{2} \qquad (1)$$

In the middle part of the window (marked with navy blue rectangle) additional case settings for FEM analysis are made. After reading *.ms2 file, tabs are filled with values and taken from *.ms2 file. These is minimum data needed for FEM analysis and can be modified for user’s specific case. Detailed description of the tabs is in the next manual section. If the user wants to omit this data, exporting only unmodified mesh from Panukl and pressure loads, „Export case settings” flag ⑦ has to be turned off.

In the lower part of the window flag for fuselage export ⑧ and type of mesh flags ⑨ are placed. In case of choosing volume mesh flag, only main wing will be exported (wing with 0 number). Additionally, for volume mesh, number of layers across wing profile height has to be specified ⑩.

Lower, four buttons are placed, with described functions:

Export - Parameters set in FEMexport are saved to *.fem file and export from Panukl to CalculiX begins.

Open File - Opens previously saved data *.fem file

Save File - Writes data set in FEMexport to *.fem file

Close - Closes FEMexport

ATTENTION!!!

By default output file for CalculiX *.inp is saved in working directory of Panukl. Panukl mesh file has the same file extension and can be overwritten!

The best solution is to set in Panukl file paths working directory of CalculiX (by default turned off) and send results directly to the place where they are analyzed.

Additional settings of FEM analysis

In the tabs of FEMexport FEM analysis case of an airplane can be defined, which will be analyzed in CalculiX. Most of the tabs contain data corresponding to data section names in CalculiX input file *.inp.

Tab CUT LE/TE

In this tab wing box geometry for FEM analysis is set, by cutting leading edge, trailing edge and adding wing box walls Fig. 2.

Data set:

- Name of wing, which will be modified
- Flag to cut or not leading edge panels
- Flag to cut or not trailing edge panels
- Number of panels to cut from the leading edge
- Number of panels to cut from the trailing edge

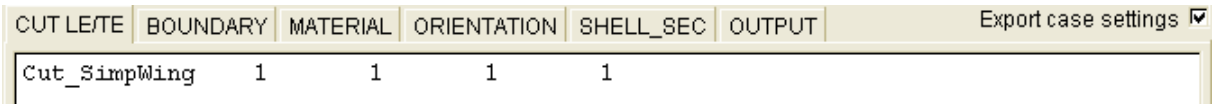

Fig. 2 Tab CUT LE/TE with definition of wing box for single wing.

Tab BOUNDARY

In this section it is possible to define boundary conditions for the wings root ribs Fig. 3. FEMexport will find nodes of the root ribs, which should be fixed.

Data set:

- Name of the wing which will be fixed
- Degrees of freedom to fix in xyz direction and rotations around three axis

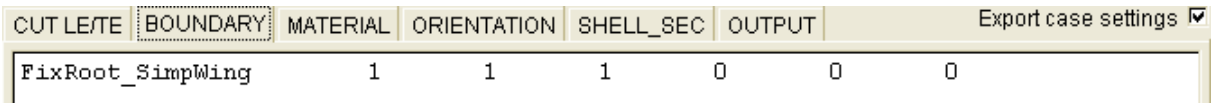

Fig. 3 Wing with fixed movement in xyz direction and with free rotation.

Tab MATERIAL

Definition of this section is the same as in material definition in CalculiX. Number of ways and types of materials can be defined. After reading *.ms2 file simple default material definition is set. Material data units should correspond to geometry and load units. It is recommended to use SI units to avoid errors! This part is copied to input file *.inp for CalculiX. More information can be found in CalculiX manual under *MATERIAL definition.

Tab ORIENTATION

Thanks to local coordinate systems directions of anisotropic materials can be set.

Ten fragment pliku jest kopiowany do pliku wsadowego do obliczeń FEM. This part is copied to input file *.inp for CalculiX. More information can be found in CalculiX manual under *ORIENTATION definition.

Tab SHELL_SEC

In the case of shell structure wing every wing will contain structure parts like WallLE, WallTE, Rib, Skin, which are defined in sub-tabs of SHELL_SEC tab. To every structure part material, material coordinate system and panel thickness is assigned. New structure elements are created on every division of mesh along wing span. During export panels are assigned to appropriate element sets representing wing structures. Numeration begins from root of the wing with “0” to the tip.

Data set:

- Name of wing and number of cross sections
- Number of section along span „y”, name of element set „ELSET”, name of material „MAT”, material coordinate system „OR” and structure panels thiskness„TH”

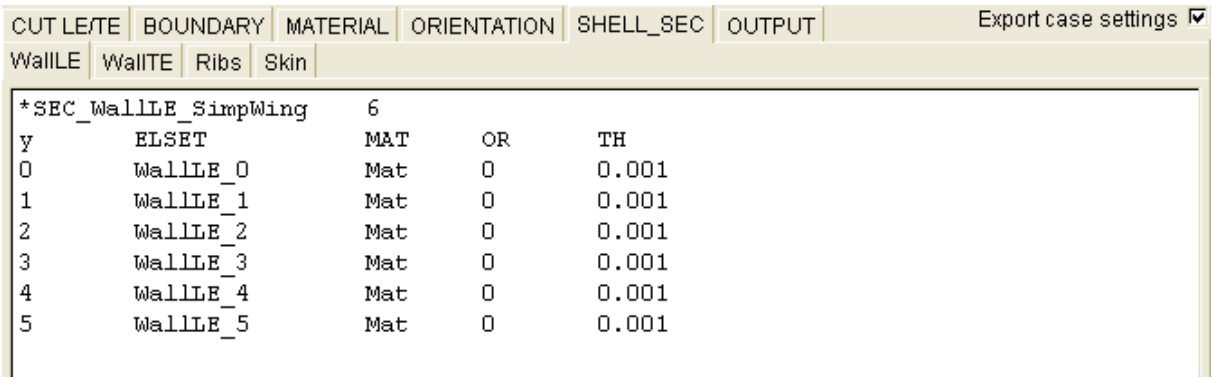

Fig. 4 Tab SHELL_SEC.

If in column „OR” instead of coordinate system name “0” value is set, than material coordinate system for the section won’t be defined. If in column “MAT” or “TH” instead of material name or thickness value “0” will be set the section won’t be created, because it won’t work in FEM analysis. In such a way for example number of ribs can be reduced (by default ribs are created in every mesh division along wing span).

Tab OUTPUT

Data in OUTPUT tab is the same as data definition in CalculiX and describes type of data output from FEM analysis in CalculiX.

Ten fragment pliku jest kopiowany do pliku wsadowego *.inp do obliczeń FEM. Więcej szczegółów można znaleźć w instrukcji do programu CalculiX. This part is copied to input file *.inp for CalculiX. More information can be found in CalculiX manual.