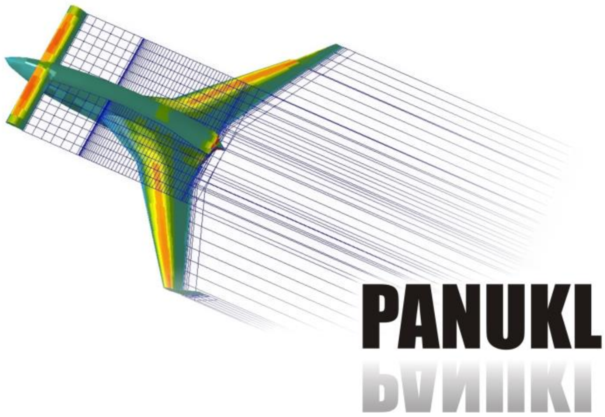

Version **2018**

Table of content

1. Introduction

FEMexport is software for structural analyses in FEM software. The computational case is defined basing on Panukl's mesh for aerodynamic analysis and pressure distribution results. The default output file is compatible with open source FEM software CalculiX, but it can be also converted to other formats suitable for example for Abacus, Ansys, Nastran, etc.

The basic mesh from Panukl can be modified, by adding internal structure components like: ribs, spars, etc. for wings and frames, stringers, etc. for fuselage. It is also possible to make FEM analysis of wings with volume elements.

Another feature is mass estimation analyses based on the definition of FEM model, where parts of mesh have assigned different material properties with differing density.

1. Manual conventions

1.1. FEMexport known limitations

- No triangle panels on fuselage skin - manual correction in out.*
- Remove parts of stringers - only by specific panels
- Floor through one stringer
- All wings have to be closed with Tip Rib
- All present wings in *ms2 have to be active
- Beam direction for wings fixed 0, 0, 1
- Don't use "_" in names of wings and fuselage
- Only 2 node beam elements (3 node not applied)

1.2. How to read the manual

- Words in *Italics* indicate options seen in the GUI.
- Parts with **Bold** words are very important.

2. FEMexport main view

The main window of the FEMexport software is shown on Figure 2.1. At the top of the window in frame *Panukl Data* input from Panukl is set. In the middle of the window number of tabs can be seen, initially only *COMPUTE* tab is active. In the *COMUTE* tab output options are set. Yellow fields indicated minimum needed data to make basic export from Panukl to FEM. That is Panukl mesh file *.inp and output file from FEMexport, which is input for CalculiX and also has extension *.inp.

ATTENTION!!!

Different programs (Panukl and CalculiX) have same file extensions. File paths and names of the files need to have unique names to avoid being overwritten during export!

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.

If the input file from Panukl and output file from FEMexport (input for CalculiX) are set correctly, configuration file *.fem of the FEMexport have to be saved *Save File* button. Saved configuration file can be read in, for example in later user session, with *Open File* button. If user tries to make export with *Export* button, before saving the configuration file, one will be prompted to save the configuration file. The result of export should be output file from FEMexport with basic mesh, which can be read by CalculiX, or viewed in any text editor.

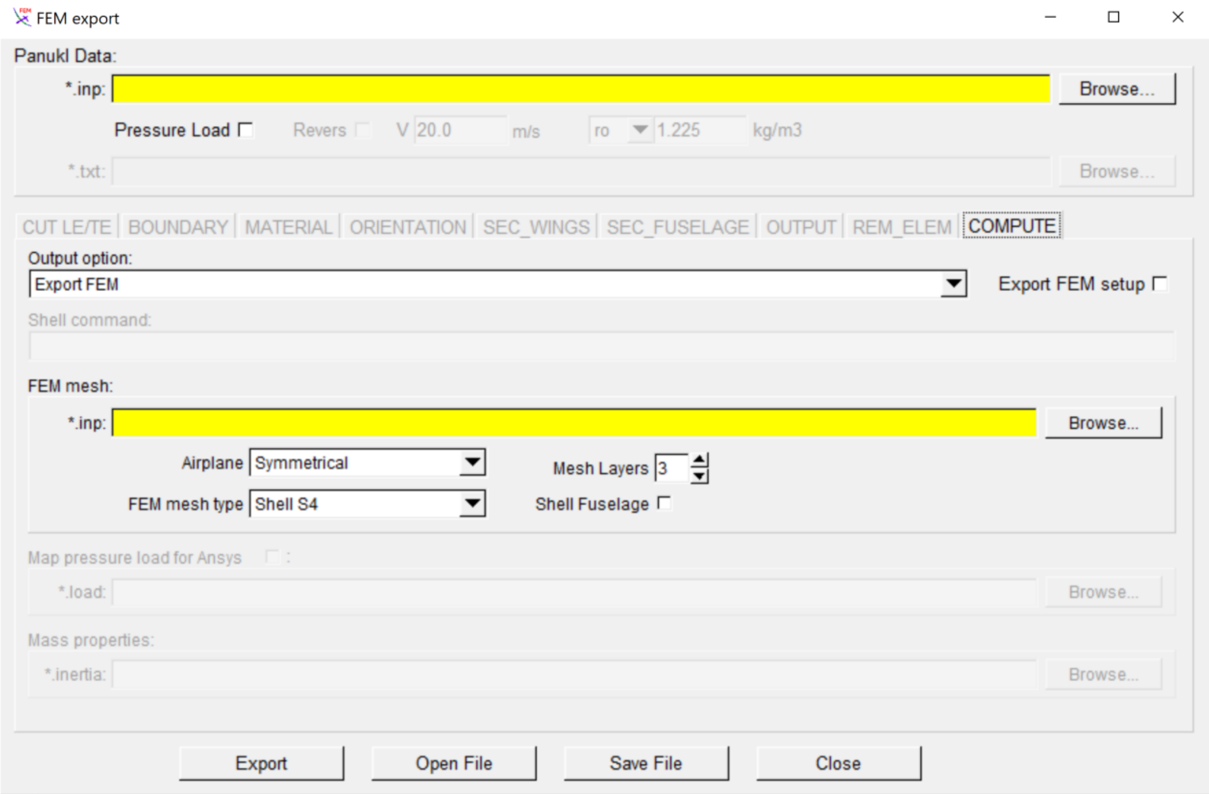

Figure 2.1 Main window of the FEMexport after opening.

2.1. Aerodynamic loads

To add aerodynamic loads from Panukl option *Pressure Load* has to be checked, which will activate additional settings for definition of the aerodynamic loads. Figure 2.2 shows GUI with the basic setup, where input and output file paths for the basic wing from Panukl examples was chosen. Additionally *Pressure Load* flag is turned on and path to file with data containing pressure coefficient distribution on panels *.txt is set.

In CalculiX software normal vector of the panels is pointing from the airplane to the free space. Pressures in Panukl apply to convention used in aerospace, pressure with minus sign sucks the wing's 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 Figure 2.2. Nondimensional pressure coefficients from Panukl has to be multiplied by dynamic pressure to obtain pressure on surfaces with units (1). This is done by setting speed of flight *V* and air density *x*. Air density *ro* can be determined explicit, or from the flight altitude above sea level *H*. From the list user can choose to export whole or only half of the airplane for FEM analysis.

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

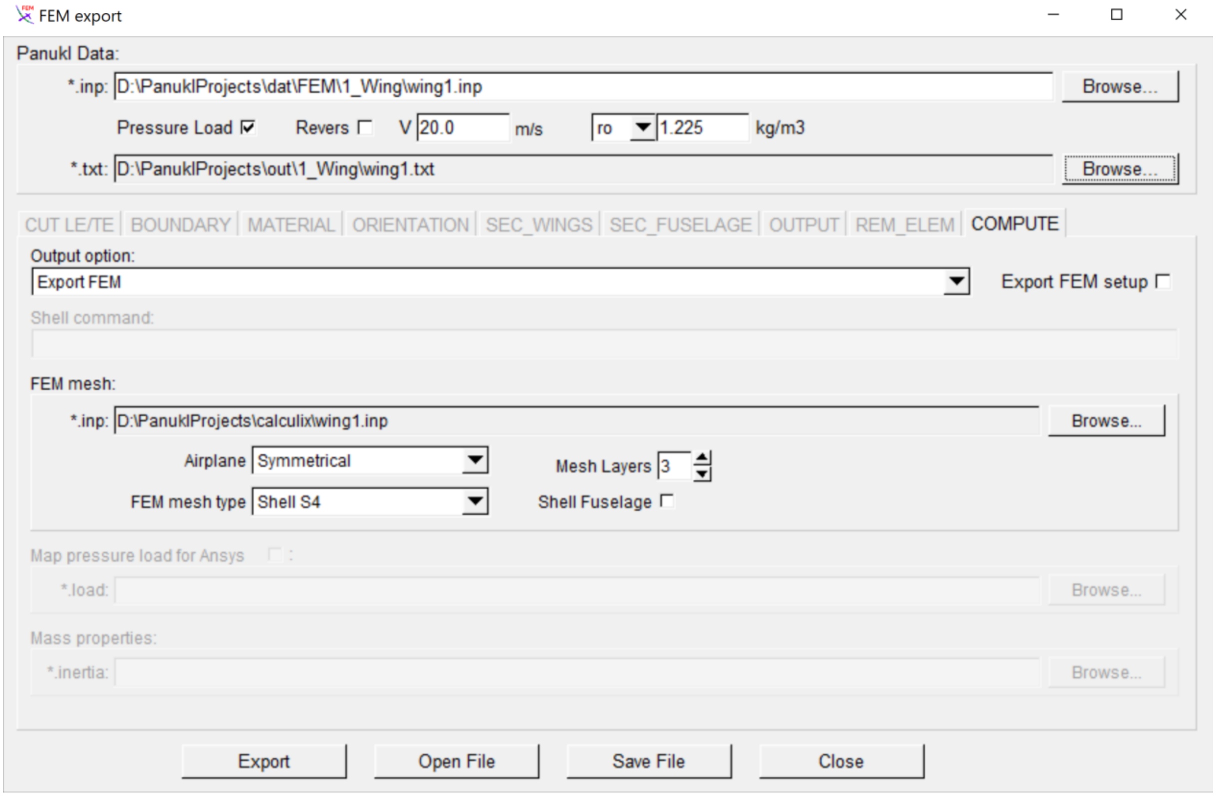

Figure 2.2 Basic export configuration with added aerodynamic loads.

2.2. Output options

Global output mesh options can be changed in the *COMPUTE* tab, *FEM mesh* frame.

- By default whole symmetrical airplane grid is exported. This can be changed by choosing different option for *Airplane* filed, available options are: *Symmetrical/Right half/Left half*
- The default *FEM mesh type* elements are 4 node shells elements *S4*. The option can be also changed for 8 node shell elements *S8*, or volume, brick elements *C3D8*. To the meshes with shell elements also beam elements can be added later in FEMexport.
- Option *Mesh Layers* indicates for how many layers ribs and spar walls inside wings will be divided.
- By default fuselage mesh is not exported, to export fuselage mesh check box for *Shell Fuselage* option.

The basic *Output option* is just creating output file from export, which is input for CalculiX. More *Output options* are available Figure 2.3. After export the results can be automatically read in to CalculiX preprocessor and displayed, analysed with CalculiX, or analysed in CalculiX and results of FEM analyses displayed in postprocessor. The least Output option enables to make structure mass analyses, which will be discussed in separate chapter.

On Figure 2.3 option *Export FEM setup* is turned on. It can be noticed that after checking the option other tabs got activated. With *Export FEM setup* option turned on the basic mesh from Panukl can be significantly changed, boundary conditions can be set, materials defined and assigned to mesh parts and more. **Tables with data to fill, or change in the tabs will activate only after reading in Panukl's mesh file.** Details of setting up extended options for FEM analyses will be discussed in the following chapters.

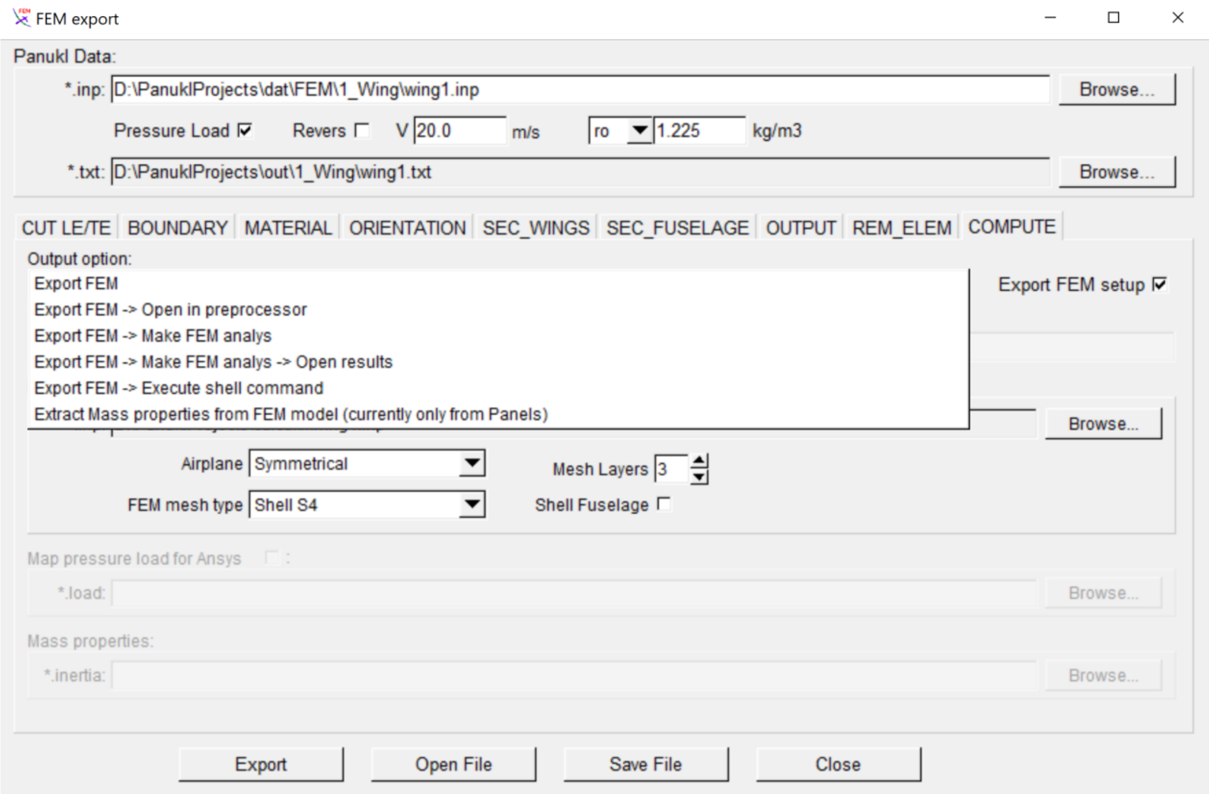

Figure 2.3 Output options.

2.3. Mapping loads for Ansys

Option *Map pressure load for Ansys* is not active yet.

2.4. Mass properties

After choosing the last options from *Output options*, which is *Extract Mass properties from FEM model*, frame with *Mass properties* will be activated. Notice that the *Export* button changed label to *Mass properties*. Output file for the resulting mass properties has to be chosen Figure 2.4. **The mass properties can be accurately computed only when whole FEM case is completely set up**, which is the subject of the following chapter.

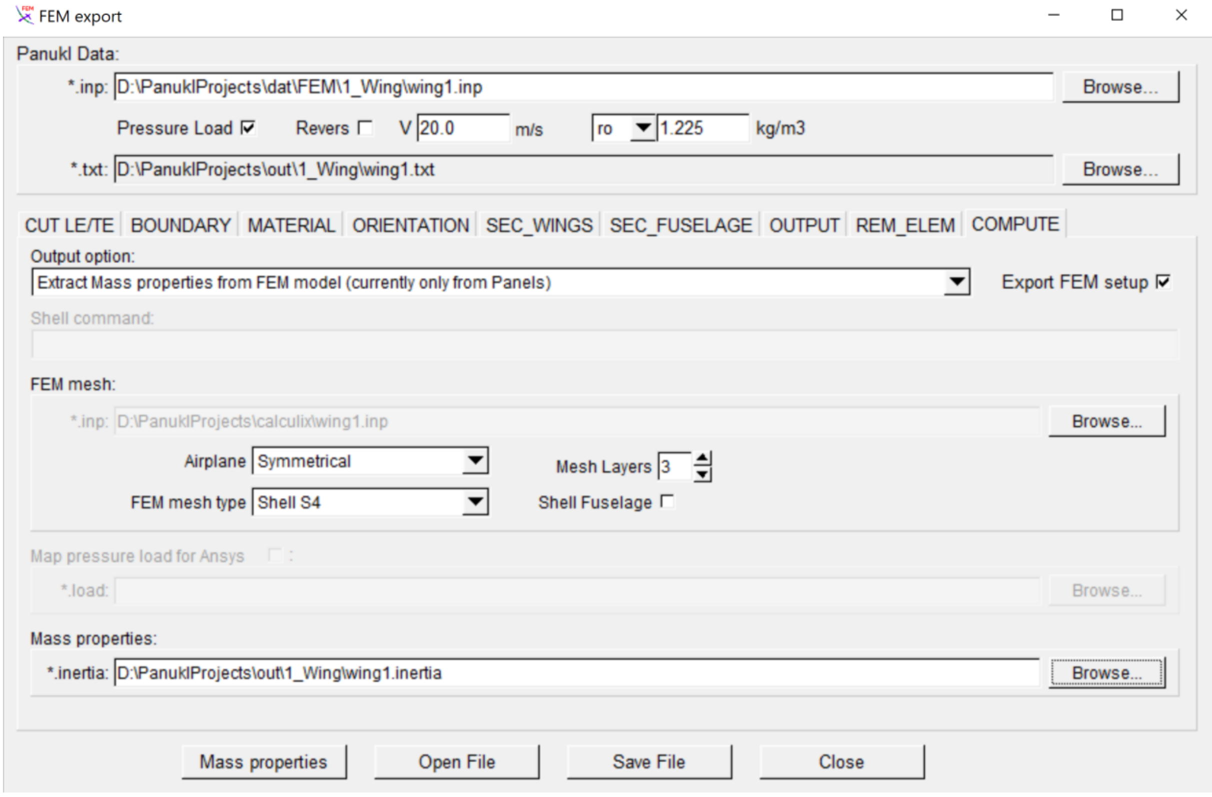

Figure 2.4 Mass properties.

3. Export FEM setup

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. **New section of panels of the structure parts is limited by the initial divisions of the grid from Panukl.**

3.1. Tab CUT LE/TE

In this tab wing box geometry for FEM analysis is set, by cutting leading edge and/or trailing edge skin panels and adding spars' walls Figure 3.1.

Data set:

- *Id* - of the wing
- *Wing* - name of the wing, which will be modified
- *LE cut* - flag to cut, or not leading edge skin panels
- *TE cut* - flag to cut, or not trailing edge skin panels

- *LE panels* - number of skin panels to cut from the leading edge,

  also placement of the wall of the front spar (even when skin panels aren't cut)

- *TE panels* - number of skin panels to cut from the trailing edge,

  also placement of the wall of the rear spar (even when skin panels aren't cut)

- *Panels* – remaining panels between front and rear spar, table fields will turn red indicating warning if the number of panels is lower than 0

- *Image* – pictogram showing how the wing's cross section will look like with the current settings

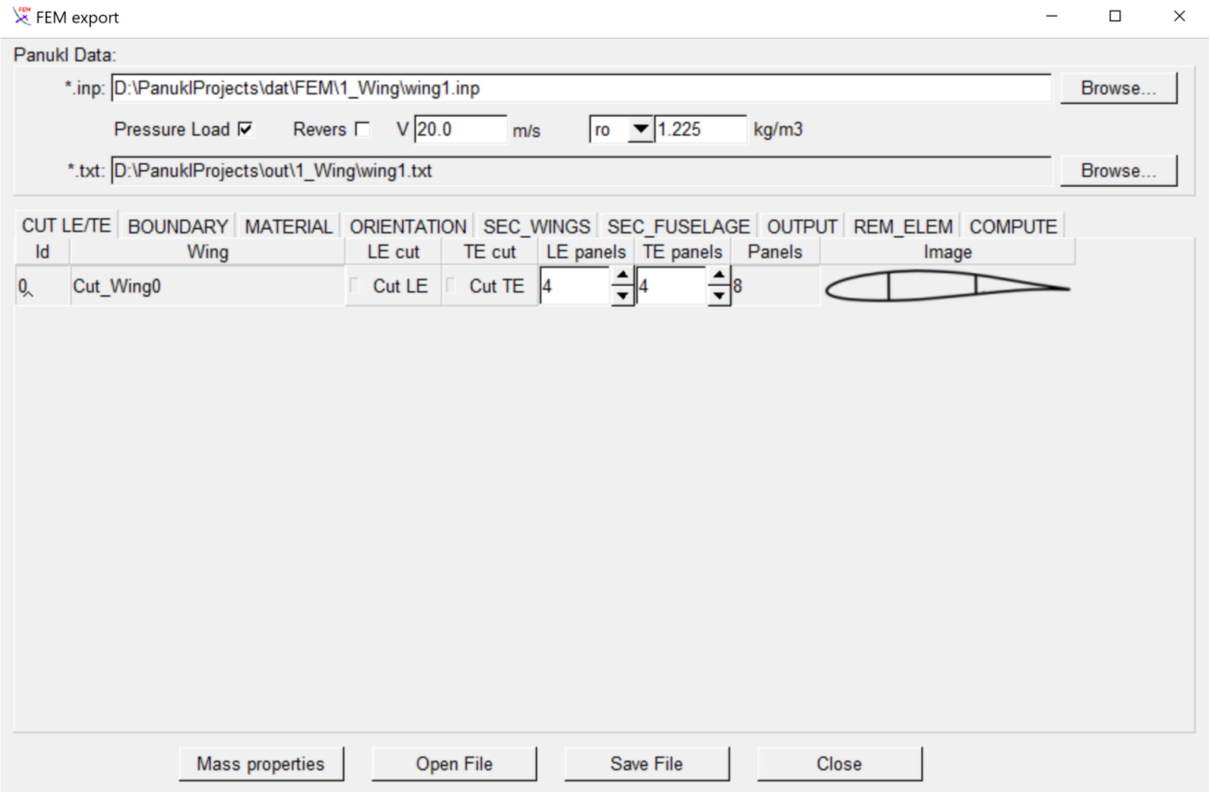

Figure 3.1 Tab CUT LE/TE with definition of wing box for single wing.

3.2. Tab BOUNDARY

In this section it is possible to define boundary conditions for the wings' root ribs Fig. 3. FEMexport finds nodes of the root ribs and the chosen degrees of freedom are fixed.

Data set:

- *Id* - of the wing
- *Wing* - name of the wing, which will be modified

- *x trans* – translation in x direction to be fixed
- *y trans* – translation in y direction to be fixed
- *z trans* – translation in z direction to be fixed
- *x rot* – rotation about x axis to be fixed
- *y rot* – rotation about y axis to be fixed
- *z rot* – rotation about z axis to be fixed

3.3. 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 aluminum alloy 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 the input file *.inp for CalculiX. More information can be found in CalculiX manual under *MATERIAL definition.

Composite Material

Orthotropic material from composite layers can be defined. The interface for the material definition Figure 3.2 is opened with the *Composite Material* button.

Data set:

- *Id* - of the composite layer
- *Ply Name* - name of the composite ply for description
- *Th* – thickness of the ply
- *fi* – angle by which the ply was rotated from the orthogonal directions
- *E1* – Young modulus in the direction one
- *E2* – Young modulus in the direction two
- *G12* – Deformation modulus
- *v12* – Poisson coefficient
- ro – density of the wet composite (used for mass analyses)

After setting up all the data and pressing *Compute Orthotropic Material* smeared properties of the composite material are computed and converted to the orthotropic material properties. Results are displayed in the *Orthotropic material* box and can be copied to the main material definition window with the *Paste* button.

With the orthotropic material it is not possible to analyse structure's separate composite layers but the material maintains anisotropic properties based on the composite layers. Time of computations with the orthotropic material is much shorter than with full composite definition. It is good compromise between results accuracy and time of computation for the preliminary design. After obtaining satisfactory results, with this approach, one can define full composite material according to the CalculiX composite material definition.

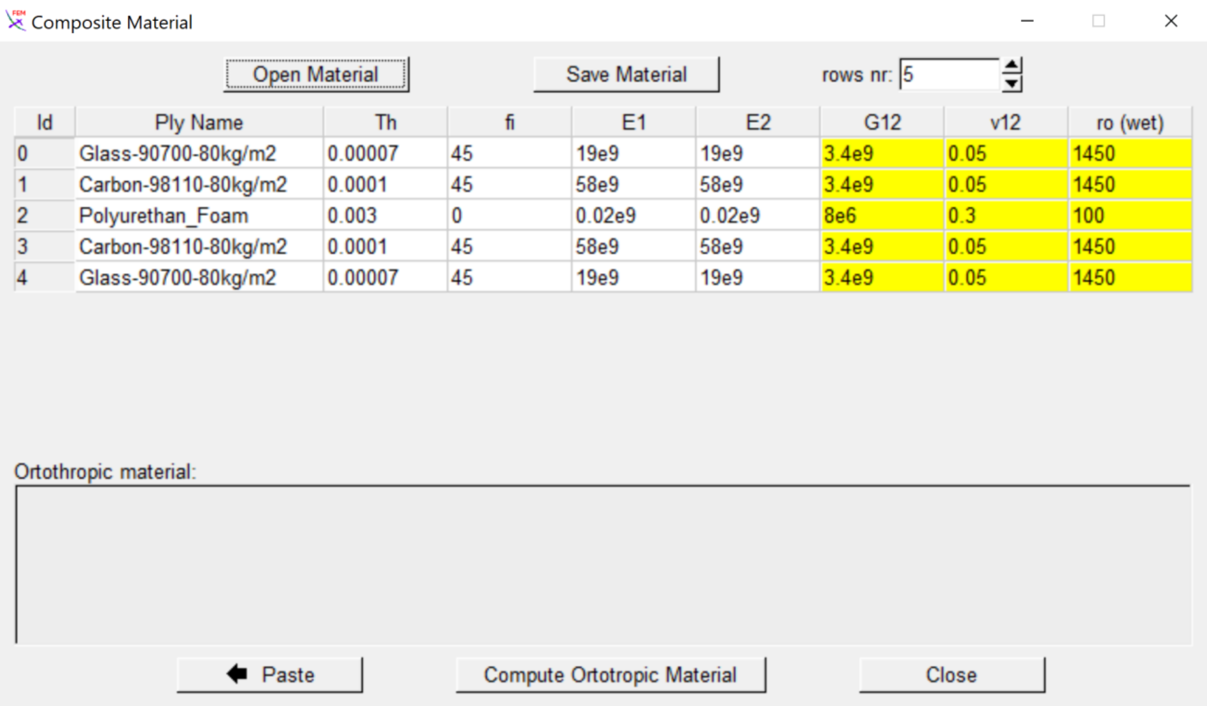

Figure 3.2 Interface for composite material definition.

3.4. Tab ORIENTATION

Thanks to the local coordinate systems directions for anisotropic materials can be set. This part is copied to the input file *.inp for CalculiX. More information can be found in CalculiX manual under *ORIENTATION definition.

3.5. Tab SEC_WINGS

In the case of shell structure every wing will contain structure parts like *SparCupsLE, SparCupsTE, WallLE, WallTE, Ribs, SkinTop, SkinBottom, Beam-Stringers* which are defined in sub-tabs of *SEC_WINGS* tab.

To every structure part (**elements set**) material, material coordinate system and panel thickness is assigned. New structure elements are created on every division of the basic mesh from Panukl, 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.

Sub-structures:

- *SparCupsLE* – selected panels of wing's skin, for which different material can be assigned to form part of **front** spar cups
- *SparCupsTE* – selected panels of wing's skin, for which different material can be assigned to form part of **rear** spar cups
- *WallLE* – selected panels for which different material can be assigned to form part of the **front** spar wall
- *WallTE* – selected panels for which different material can be assigned to form part of the **rear** spar wall
- *Ribs* – wing's ribs for which different material can be assigned
- *SkinTop* – selected panels from wing's **top** skin for which different material can be assigned
- *SkinBottom* – selected panels from wing's **bottom** skin for which different material can be assigned
- *Beam-Stringers* – beam stringers properties for the whole wing (optional)

Data set of substructures:

- *Id* - of the part of the substructure (**elements set**)
- *ELSET* - name of the particular **elements set**, which will appear in CalculiX (see also CalculiX manual)
- *MAT* - material name assigned to the **elements set** (see also CalculiX manual)
- *OR* – optional local coordinate system for the **elements set** (see also CalculiX manual)
- *TH* – thickness of the elements assigned to the **elements set** (see also CalculiX manual)
- *F Panels* – number of panels assigned to the **elements set** in front of the wall of the spar **(options applies only to the substructure of the spar cups)**
- *R Panels* – number of panels assigned to the **elements set** after the wall of the spar **(options applies only to the substructure of the spar cups)**

ELSET cells in the table of the wing are marked with a light blue color. If there are more wings in the model, than subsequent ELSET names of the other wings will appear in the next rows of the table, **but will be indicated with different shades of the blue color**.

If in the column *OR* instead of coordinate system name value 0 is set, than the coordinate system for the section won't be defined. **If in the column *MAT*, or *TH* instead of material name, or thickness value 0 will be set the section won't be created. In those cases color of the cells in the table changes to orange Figure 3.4. In such a way for example number of ribs can be reduced, or any other type of the internal sub-structure which is not needed (by default ribs are created in every mesh division along wing span).**

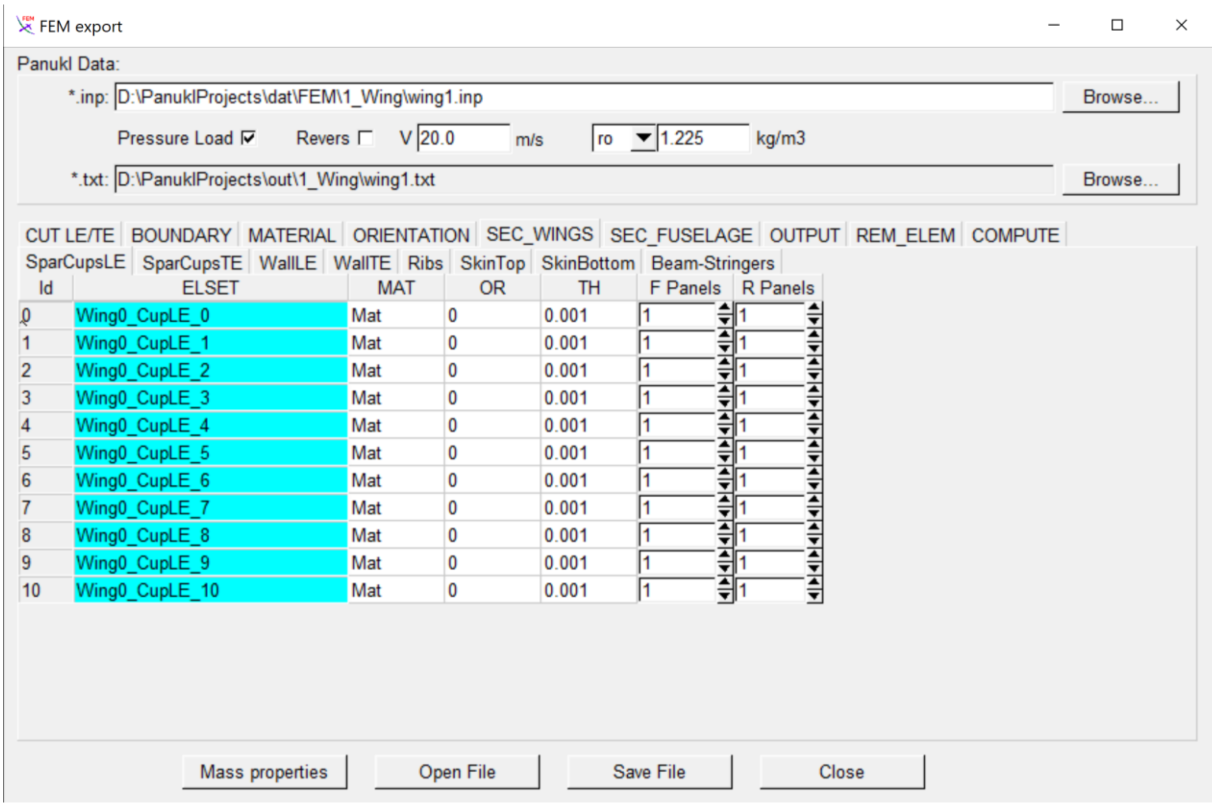

Figure 3.3 Wings sections.

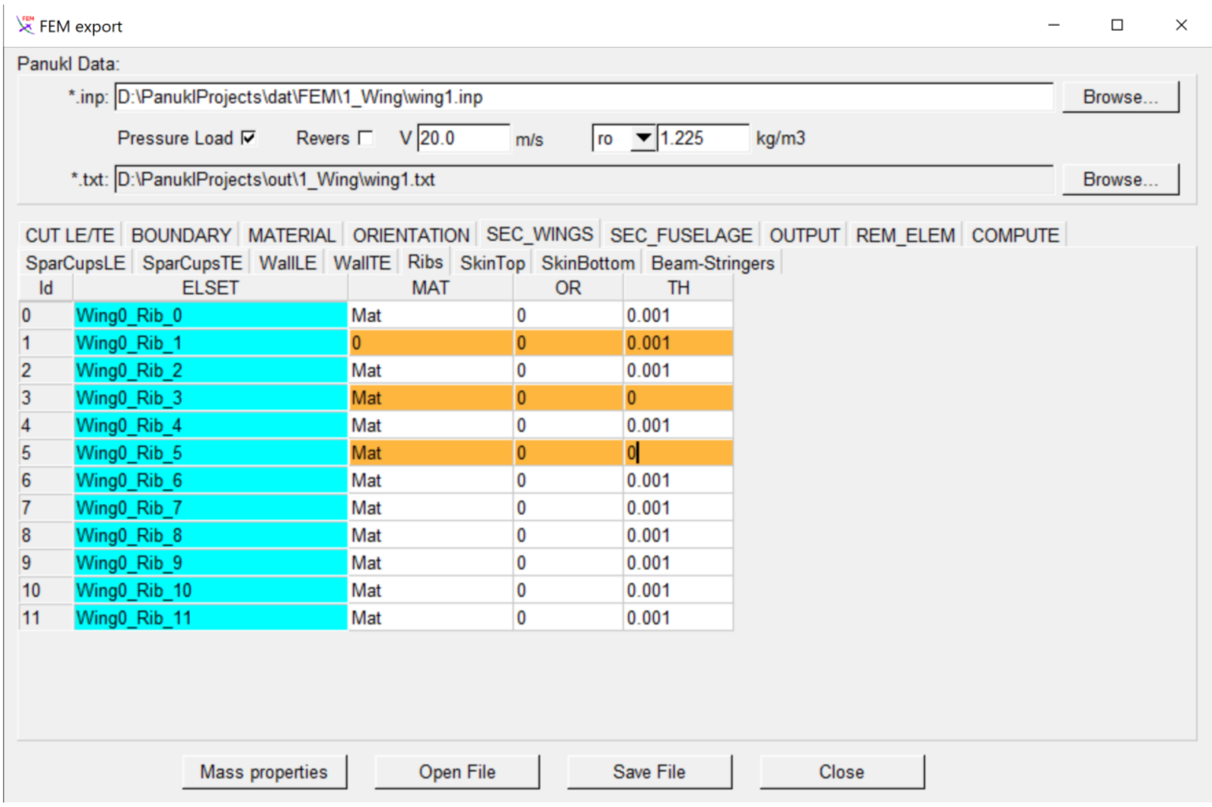

Figure 3.4 Wings sections – with few ribs turned off.

Beam-Stringers

Wing can have additional beam elements representing span wise stringers. Every pair of stringers (on the top and bottom surface of the wing) is generated at the chord wise division of the basic mesh from Panukl (excluding LE and TE). **By default material of every stringer is 0, so they are not active**. Figure 3.5 shows defined stringers between wing's spars, by assigning material name.

Data set of substructures:

- *Id* - of the part of the substructure (**elements set**)
- *ELSET* - name of the particular **elements set**, which will appear in CalculiX (see also CalculiX manual)
- *MAT* - material name assigned to the **elements set** (see also CalculiX manual)
- *OR* – optional local coordinate system for the **elements set** (see also CalculiX manual)
- *TH1* – first thickness of the beam elements assigned to the **elements set**, (direction of thickness definition is **perpendicular to the wing surface**) (see also CalculiX manual)
- *TH2* – second thickness of the beam elements assigned to the **elements set**, (direction of thickness definition is **parallel to the wing surface**) (see also CalculiX manual)

- *Shape* – available beam cross section shapes in CalculiX - **Rectangular, Circular/Eliptical (circular cross sections work only with 3 node beam elements, not applicable yet in FEMexport)** (see also CalculiX manual)

- *Offset1* – offset of the beam center from nodes defining the beam, measured in the fraction of the thickness *TH1* in the considered direction (direction of thickness definition is **perpendicular to the wing surface**) (see also CalculiX manual)

- *Offset2* – offset of the beam center from nodes defining the beam, measured in the fraction of the thickness *TH2* in the considered direction (direction of thickness definition is **parallel to the wing surface**) (see also CalculiX manual)

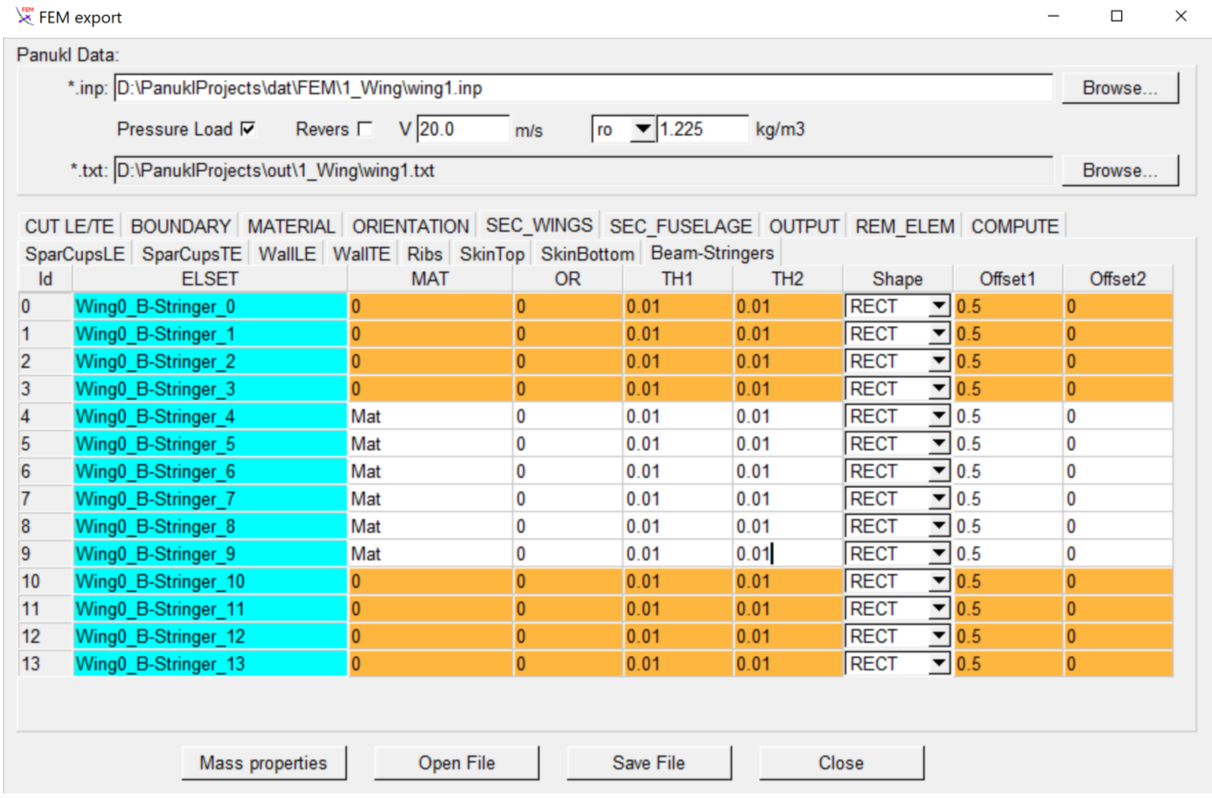

Figure 3.5 Wing beam stringers.

3.6. Tab SEC_FUSELAGE

In the case of shell structure fuselage will contain structure parts like *Skin, Frames, Stringers, Beam-Stringers* which are defined in sub-tabs of *SEC_FUSELAGE* tab.

To every structure part (**elements set**) material, material coordinate system and panel thickness is assigned. New structure elements are created on every division of the basic mesh from Panukl, along fuselage length and cross section division for stringers. During export panels are assigned to appropriate element sets representing fuselage structures. Numeration begins from nose of the fuselage, or bottom stringer with 0.

Sub-structures:

- *Skin* – selected panels of fuselage skin along fuselage length, for which different material can be assigned
- *Frames* – panels of fuselage frames along fuselage length for which different material can be assigned
- *Stringers* – panels of fuselage stringers at every fuselage cross section division for which different material can be assigned
- *Beam-Stringers* – beam stringers at every fuselage cross section division for which different material can be assigned (optional)

General settings:

Panel with the general settings for the fuselage sub-structures is shown on Figure 3.6.

- *Height of Frames* – height for all frames in the fuselage
- *Height of Stringers* – height for all stringers in the fuselage
- *Nodes* – number of nodes on the hight of frames, or stringers respectively
- *Floor Stringer* – number of the stringer which will be extended to the center of the fuselage to create optional floor inside of the fuselage, it also **defines z coordinate of the center of the frames, which is taken from the z coordinates of the *Floor Stringer***
- *Additional Nodes to the Center of the Frames* – used if bulkhead (full frame), or floor is created, which extend from fuselage surface to the center nodes

Data set of substructures:

Panel and beam element sets are defined in the same way as for the wings. **See section Tab SEC_WINGS for the details. The only exception is option *Partition* to choose in *Frames* table. *Frame*, or full *Bulkhead* option can be chosen.**

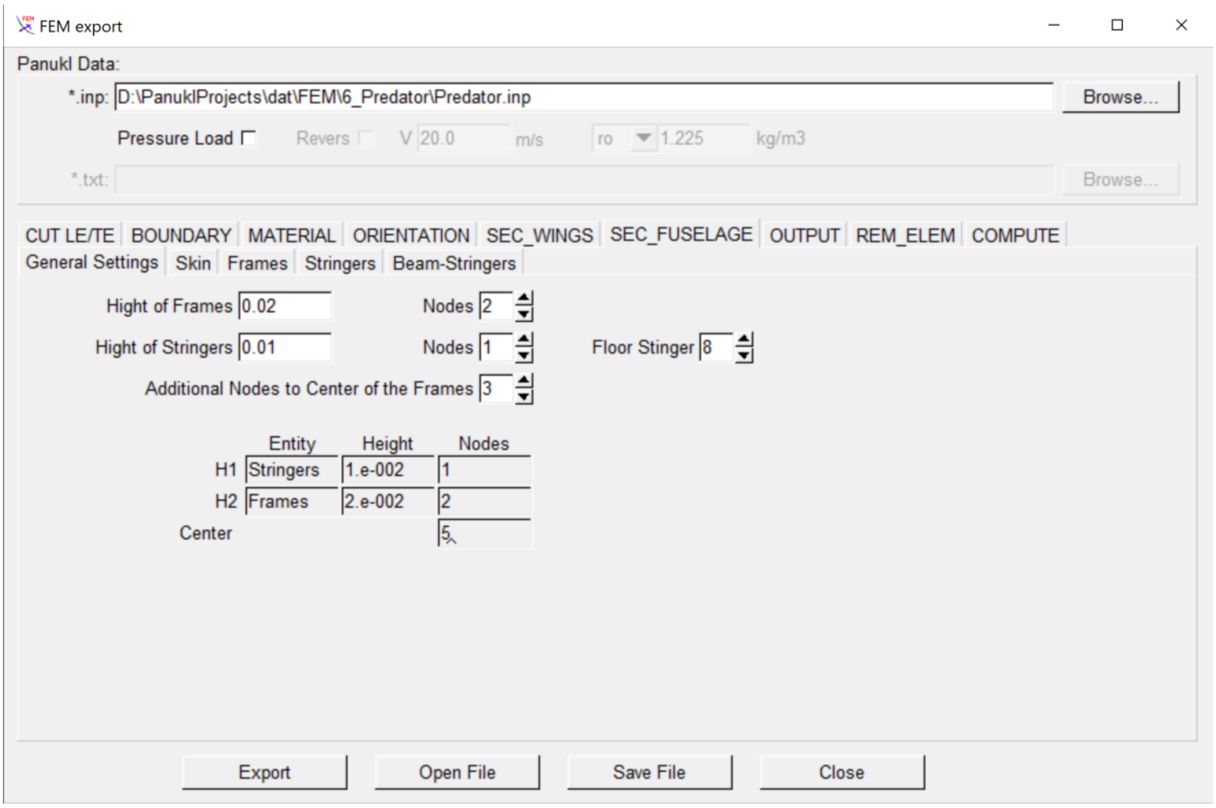

Figure 3.6 Fuselage general settings.

3.7. 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.

This part is copied to input file *.inp for CalculiX. More information can be found in CalculiX manual.

3.8. Tab REM_ELEM

In this tab number of elements can be defined to be removed from the model. It can be useful to define open cuts in the structure, or remove particular elements, which can't be turned off, by disabling whole *ELSET*. **Expression *REM_ELEM should remain for FEMexport data part recognition and processing. Numbers of the element to be removed should be separated by white spaces (backspace, tab, new line). Sequence of elements can be removed by putting first and last element of the sequence in the brackets. Selection of the elements should be done at the end, since other structure mesh modifications will change elements numbering.** Example of defining few elements to be removed is shown on Figure 3.7.

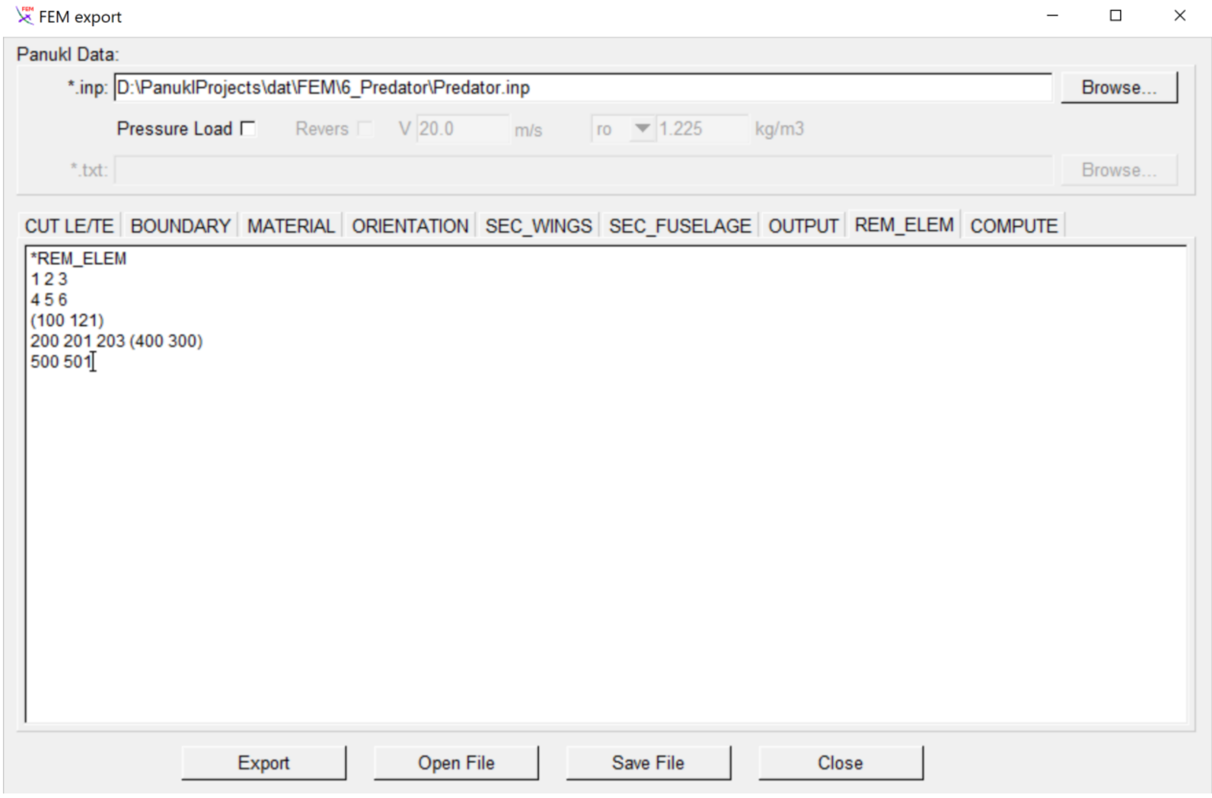

Figure 3.7 Removing elements from the model.

4. How to use FEMexport effectively

Start from basic case and then add modifications. Export process is rather quick, but usually many attempts are needed before user sets the final, desired export configuration. If error occur at any stage of the FEMexport process, try to avoid them before proceeding further.

1. Open FEMexport and set input file with grid from Panukl and output file. Save the configuration in the configuration file *.fem and make export. If the output file appeared and log file (next section) ends up with *Success!!!* modifications of the export case can be started.
2. Add aerodynamic load file and all related settings to it. Change *Output option* to *Export FEM -> Open in preprocessor* to quickly investigate visually results of the export.
3. Change global settings as desired of the *FEM mesh*, that is *Airplane Symmetry* option, *FEM mesh type,* number of *Mesh Layers,* include *Shell Fuselage*.
4. If no problems occurred, check *Export FEM setup* flag and make further, detail changes to the export case configuration in the tabs.

5. Debugging the export process

About many simple errors user is warned with window messages. More detail information can be found in the log files. Log files of the FEMexport (export GUI) and Pan2FEM (export processor) and all other Panukl programs are stored in the hidden catalogue **.panukl. The catalogue .panukl is placed under Windows in C:\Users\UserName and under Linux in the home directory. Successful export should end up with *Success!!!* statement at the end of the log Pan2FEM.txt file. Otherwise, reasons of errors and abandoning export process can be found in the log file. Usually the log ends in the place where the export was interrupted, so look for the errors especially at the end of the log.**