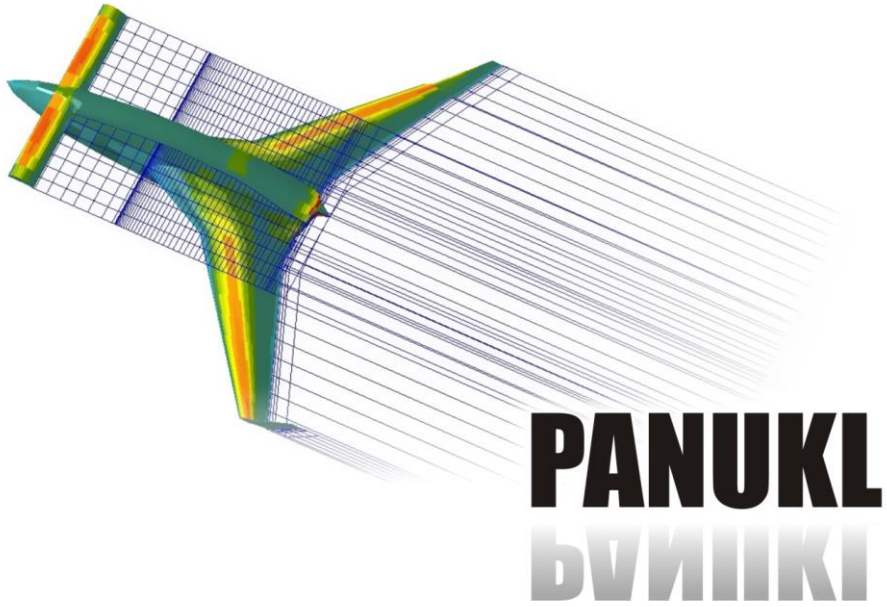

Version 2018

Table of content

PANUKL – user manual

The current document presents the description of graphic user interface of the **PANUKL 2018** package and all available options. The external programs are described in the separated files. To run **PANUKL's GUI**, click **GridView** managing subprogram icon on the desktop or in the START MENU. After few seconds the main application window displays (Fig. 1) and **PANUKL** is ready to operate.

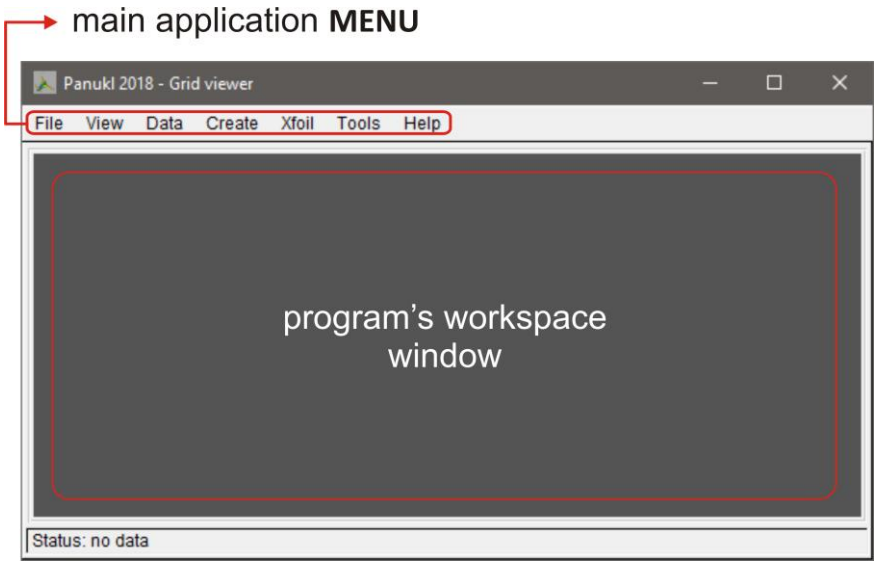

Fig. 1 - Main application window

To change program window size use standard WINDOWS buttons. User can access the particular program functions from drop down managing application MENU..

FILE menu description

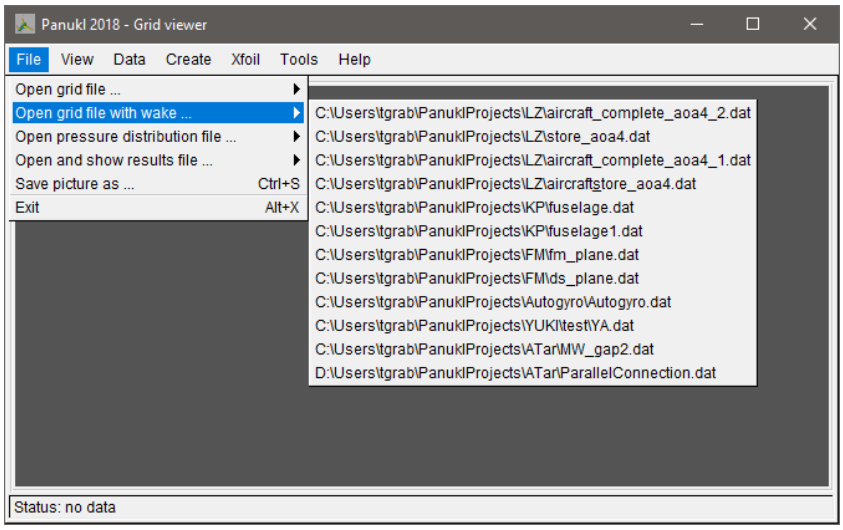

Fig. 2 - Menu – FILE

Available options in FILE menu

Function	Description
Open grid file [Ctrl+O]	Open grid file [.inp], from user selected disc location Fig. 3.
Open grid file with Wake [Alt+Ctrl+O]	Open [.dat] file from user selected disc location containing grid and wake
Open pressure distribution file [Alt+Ctrl+T]	Open [.txt] file from user selected disc location containing pressure distribution and other results (velocity components, Mach number, etc.).
Open and show results file	Show results file **[.out]** or **[.czy]**) , Fig. 5.
Save picture as [Ctrl+S]	Save the current window picture to **JPEG** , **PNG** or **BMP** format file. The dialog window will appear Fig. 4
Exit [Alt+X]	End program, exit application.

The names of last opened files (20 in each of the four groups "Open") are saved and user can select them from drop down menu (Fig. 2).

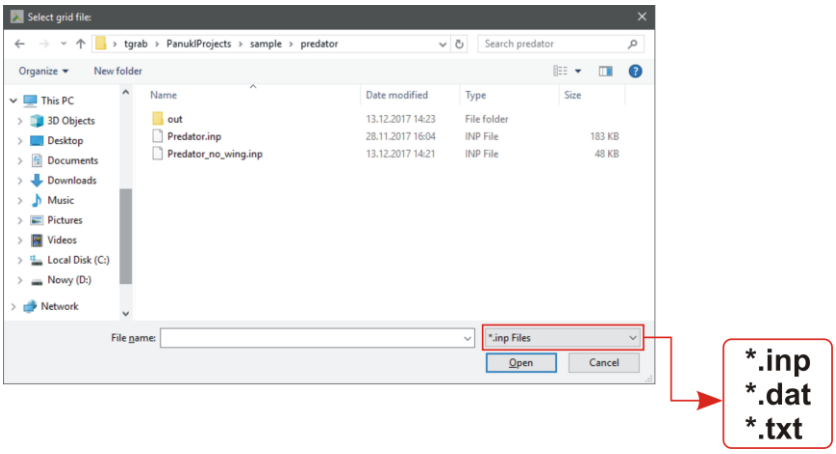

Fig. 3 – File selection window (example)

File selection Windows can be different dependent on the current operating software version, window looks does not influence PANUKL's functionality.

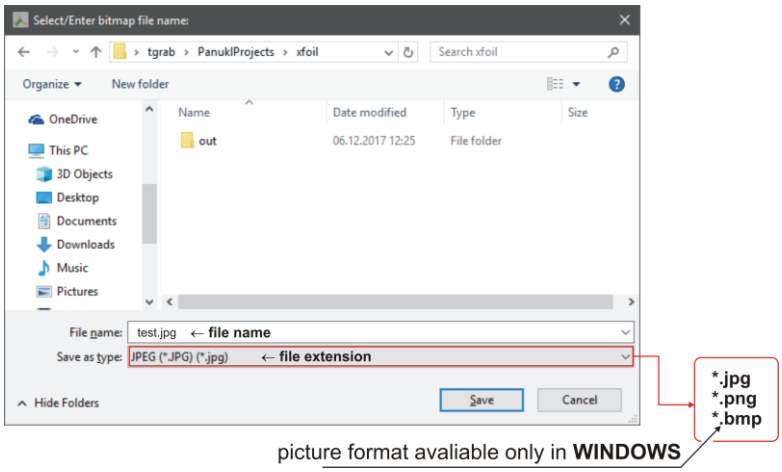

Fig. 4 – Save to graphic file current PANUKL window (only workspace)

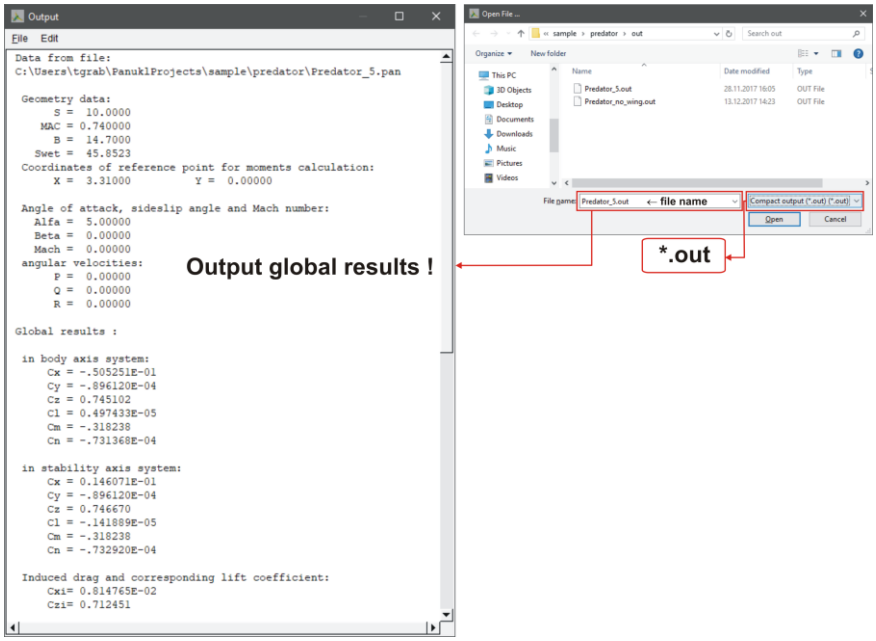

Fig. 5 – Output results window example

VIEW menu description

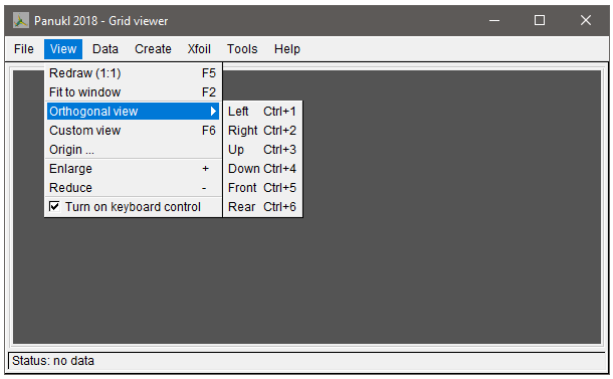

Fig. 6 – Menu – VIEW

Available options in VIEW menu

Function	Description
Redraw (1:1) **[F5]**	Redraw current object in main application window, restore the original scale of the object.
Fit to window **F2**	Fit to the window size
Orthogonal view	Orthogonal view selection: Left – Ctrl-1 Right – Ctrl-2 From up – Ctrl-3 From bottom – Ctrl-4 Front – Ctrl-5 Rear – Ctrl-6
Custom view **F6**	Saved view
Origin …	Show the window to set the origin of rotation (Fig. 7)
Enlarge **[+]**	Zoom (+)
Reduce **[-]**	Zoom (-).
Turn on keyboard control **[check box]**	Turn on keyboard control for analyzed object, Fig. 8

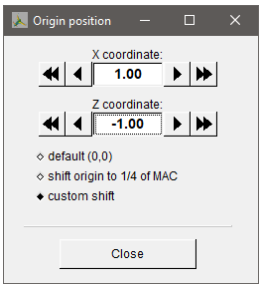

Fig. 7 – Window to set the origin of rotation

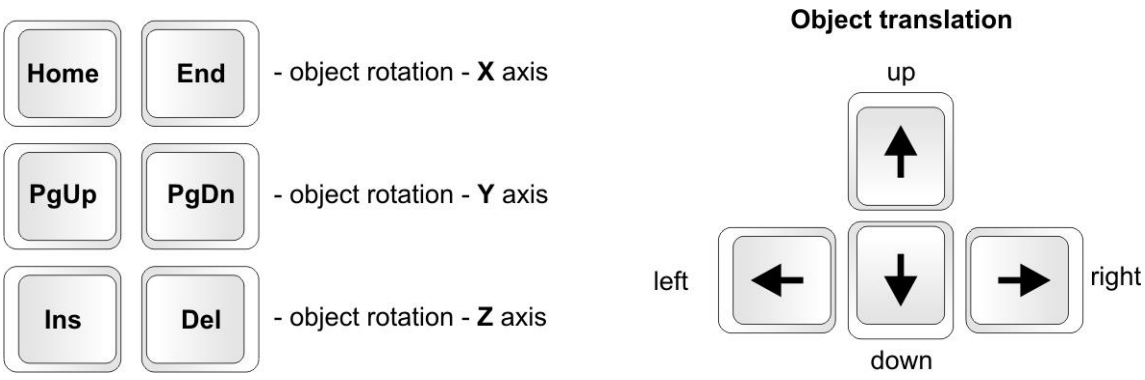

Fig. 8 – Keyboard controls explanation

DATA menu description

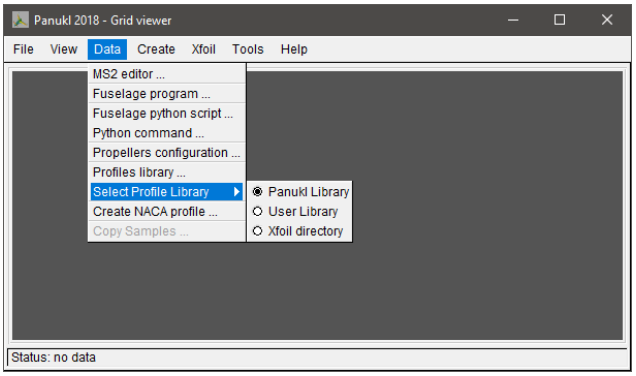

Fig. 9 – Menu - DATA

Various tools for editing and data management are available in the **DATA** menu. The following options mean:

- MS2 Editor – invoking the MS2edit program for editing MS2 script files [1]
- Fuselage program – invoking the Fuselage program utilized to create a file with the geometry of the fuselage created earlier in the program NX [2]
- Fuselage python script – invoking the Python script utilized to create a file with the geometry of the fuselage created earlier in the program NX [2]
- Python command – Python interpreter command options
- Propellers configuration – invoking the ActuatorDisc program for editing the propellers data file ([.acd]).

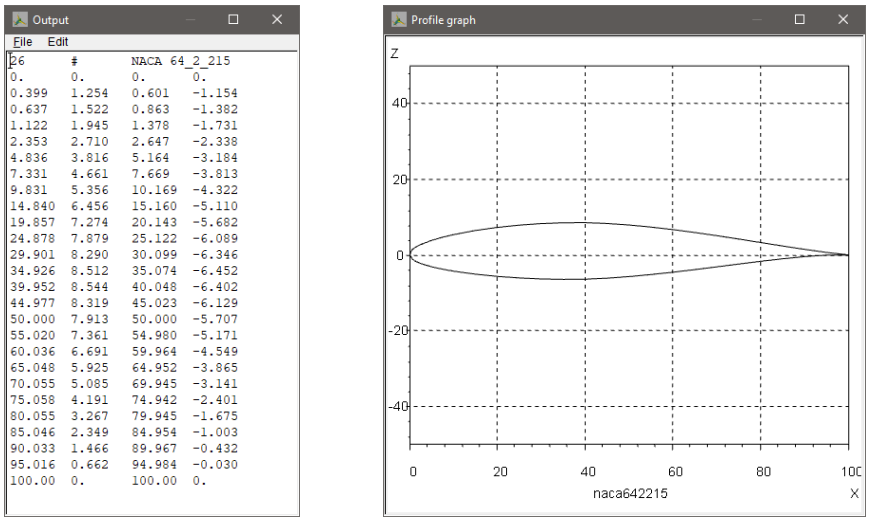

Fig. 10 - Browsing the profile library

- Profiles library – browsing the profile library (Fig. 10)
- Select Profiles library – the user has one of three options to choose from:
  - Panukl Library – library embedded in the PANUKL package
  - User Library – user profile library
  - Xfoil Library – library of profiles created/modified by the program Xfoil [3]
- Create NACA profile – create a file with profile geometry from a four or five-digit family of classic profiles NACA
- Copy Samples – copying files with examples provided with the package to the working directory. When copying, the configuration files are modified to adapt the data to the user's directory layout.

CREATE menu description

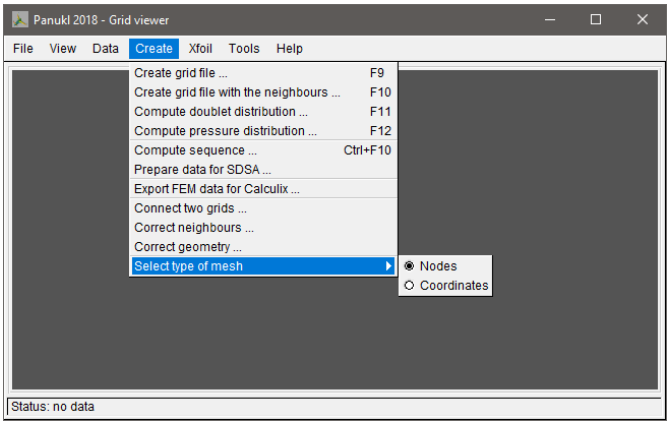

Fig. 11 – Menu – CREATE

In the CREATE menu, the user will find the most important functions of the PANUKL package that enable a complete calculation session to be performed for a given input data set.

Create grid file (F9)

The command calls the **Mesh3** program, component of the **PANUK**L package, which for a selected data set containing the description of the object (aircraft) geometry (file [.ms2]) will generate a file containing the [.inp] grid defining the object's geometry. The grid consists of quadrangular panels, Fig. 12.

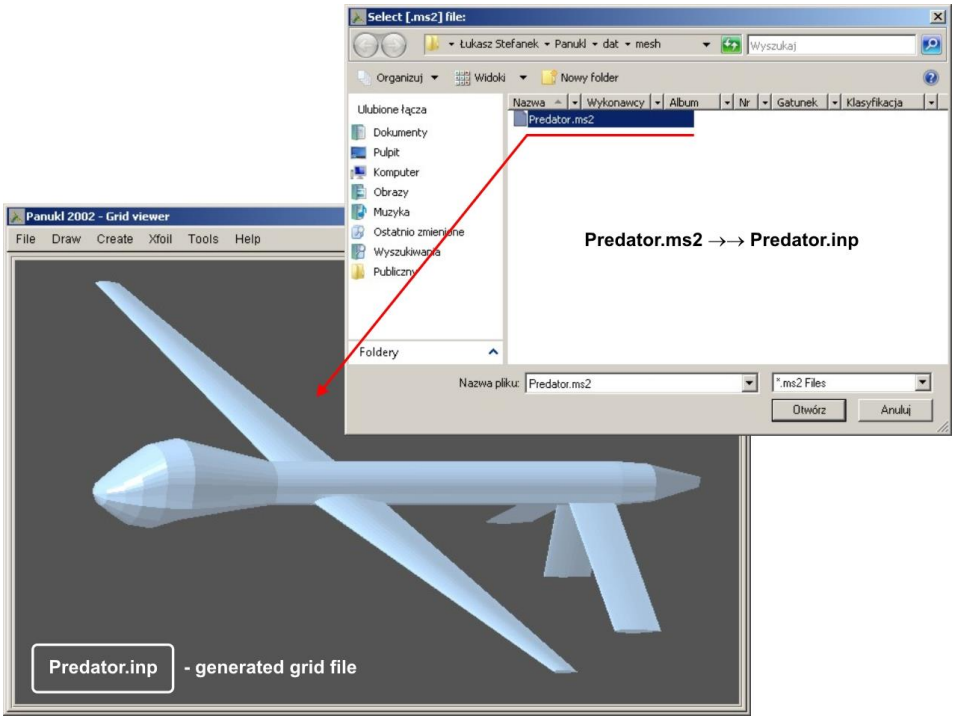

Fig. 12 – Creating grid file for current input geometry data

Create grid file with the neighbours (F10)

The command calls the Neigh program, component of the PANUKL package, which for the selected file containing the [.inp] grid of the model, will generate a [.dat] file. The output file [.dat] stores the model's grid with the generated vortex wake and information about the numbers of "neighbors" of individual grid panels

After selecting this option, the user has two options:

Option No. 1 – we do have saved on disk configuration file **[.ngh]**, Fig. 13

Run **Create grid file with the neighbours** and select saved configuration file **[.ngh]** – file contains all necessary information to create **[.dat]** file. To open selected **[.ngh]** file click **OPEN** button. Configuration window will appear (Fig. 13) where one can see saved **[.dat]** file creation options. To generate **[.dat]** file click **Save and Compute (OK)** button.

Option No. 2 – we do not have saved on disk configuration file **[.ngh]**, Fig. 13

Run **Create grid file with the neighbours** and click **CANCEL** button when prompted for saved configuration file **[.ngh]**. The configuration window will appear (Fig. 13) where user can select options to create **[.dat]** file. To save current **[.dat]** options to **[.ngh]** file click **Save [.ngh] file as**, to create **[.dat]** file click **Save and Compute (OK)** button.

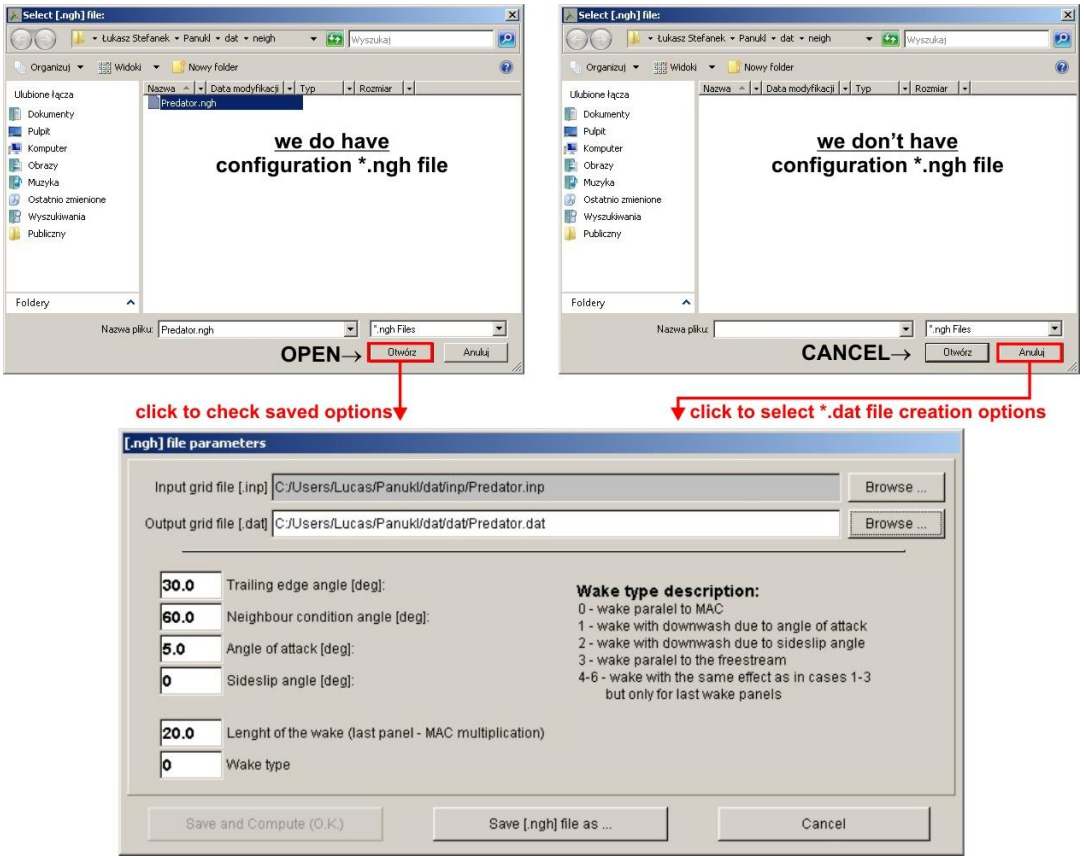

Fig. 13 – The dialog window to run Neigh program

Setting	Description
Length of the wake (MAC multiplication)	Length of the wake (MAC multiplication)
Wake type description	**Wake type creation methods:** **0** – Wake parallel to MAC **1** – Wake with downwash due to angle of attack **2** – Wake with downwash due to sideslip of attack **3** – Wake parallel to the free stream **4 , 5, 6 –** Wake with the same effect as in cases 1-3 but only for last wake panels
Trailing edge angle [deg]	Trailing edge angle. If the angle between two trailing edge grid panels is lower or equal to defined value, than wake line will be created from such trailing edge.
Neighbour condition angle [deg]	Neighbor condition angle **.** If the angle between two neighboring grid panels is higher than defined value, both panels are not treated as neighbours.
Angle of attack [deg]	Angle of attack (taken into account during wake creation).
Sideslip angle [deg]	Sideslip angle (taken into account during wake creation).

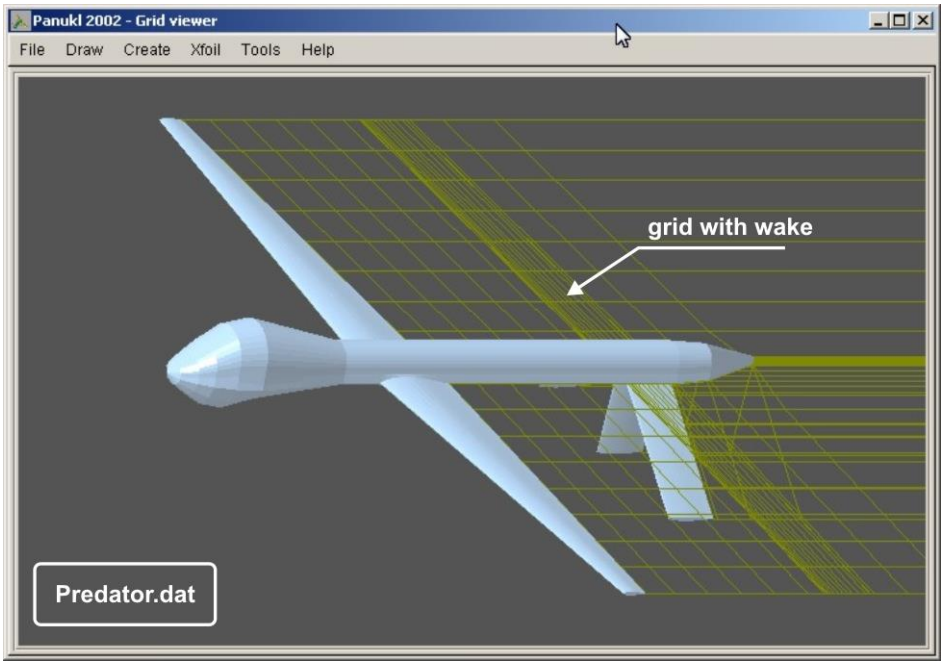

Fig. 14 – Example file: grid & wake „Predator.dat”

Compute doublet distribution (F11)

The command starts the program Panukl, component of the PANUKL package, which will generate the file [.pan] for the selected data file [.dat]. In the output file [.pan] the resultant distribution of the velocity potential is saved. The file is saved as so-called free format - read and write only by package programs.

After selecting this option, the user has two options:

Option No. 1 – we do have saved on disk configuration file **[.par]**, Fig. 15

Run **Compute doublet distribution** and select saved configuration file **[.par]** – file contains all necessary information to create **[.pan]** file. To open selected **[.par]** file click **OPEN** button. Configuration window will appear (Fig. 15) where one can see saved **[.pan]** file creation options. To generate **[.pan]** file click **Save and Compute (OK)** button.

Option No. 2 – we do not have saved on disk configuration file **[.par],** Fig. 15

Run **Compute doublet distribution** and click **CANCEL** button when prompted for saved configuration file **[.par]**. The configuration window will appear (Fig. 15) where user can select options to create **[.pan]** file. To save current **[.pan]** options to **[.par]** file click **Save [.par] file as**, to create **[.pan]** file click **Save and Compute (OK)** button.

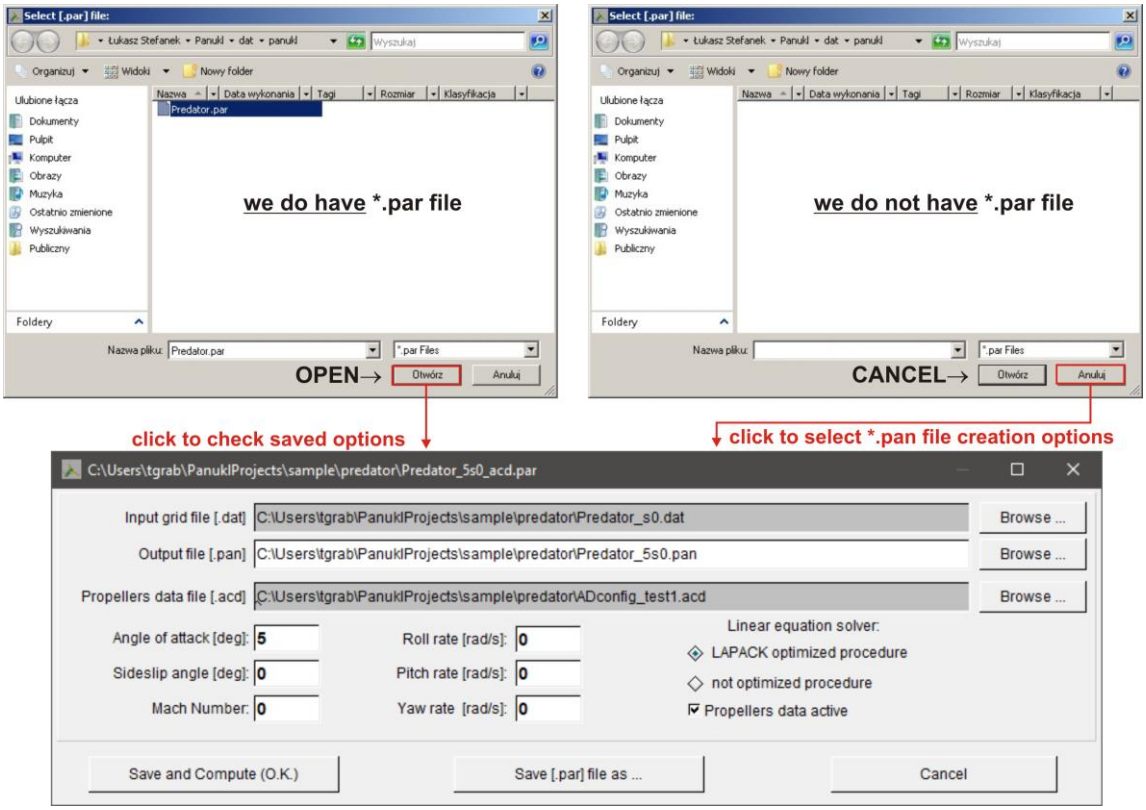

Fig. 15 – Dialog window to set parameters and to run Panukl program

Parameter/setting	Description
Linear equation solver	Linear equation solver selection: - **LAPACK optimized procedure** (default) - **not optimized procedure** (more time consuming procedure but more accurate)
Angle of attack	Angle of attack [deg], measured from free stream velocity direction and **OX** axis**.**
Sideslip angle	Sideslip angle [deg].
Mach Number	Mach number
Roll rate	**P** – roll rate [rad/s]
Pitch rate	**Q** – pitch rate [rad/s]
Yaw rate	**R** – yaw rate [rad/s]
Propellers data active	Activates the influence of the propellers system, defined in [.acd] file

The Panukl program calculates the influence coefficients and solves the system of equations resulting in the distribution of the velocity potential. This module has the highest hardware requirements. The time of calculations can be significant and depends approximately on the third power of the number of panels. After completing the calculations, the Panukl program displays the message (Fig. 16).

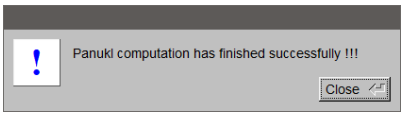

Fig. 16 – Message that computation is completed

Compute pressure distribution (F12)

The command runs the Press program, component of the PANUKL package, which for the selected data file [.pan], will generate final result files. The output file [.out] stores the results of the calculation of the total aerodynamic coefficients. The output file [.txt] stores the results of calculations related to the speed components, singularities, pressures etc. for individual panels. In the output file [.eps], the results of downwash calculation (set created optionally) are saved.

Option No. 1 – we do have saved on disk configuration file **[.prs]**, Fig. 17

Run **Compute pressure distribution** and select saved configuration file **[.prs]** – file contains all necessary information to create output result files – ***.out**, ***.txt**, ***.eps**. To open selected **[.prs]** file click **OPEN** button. Configuration window will appear (Fig. 17) where one can see saved **[.prs]** file creation options. To generate **output** files click **Save and Compute (OK)** button.

Option No. 2 – we do not have saved on disk configuration file **[.prs],** Fig. 17

Run **Compute pressure distribution** and click **CANCEL** button when prompted for saved configuration file **[.prs]**. The configuration window will appear (Fig. 17) where user can select options to create **output** files – ***.out**, ***.txt**, ***.eps**. To save current **output** files options to **[.prs]** file click **Save [.prs] file as**, to create **output** files click **Save and Compute (OK)** button.

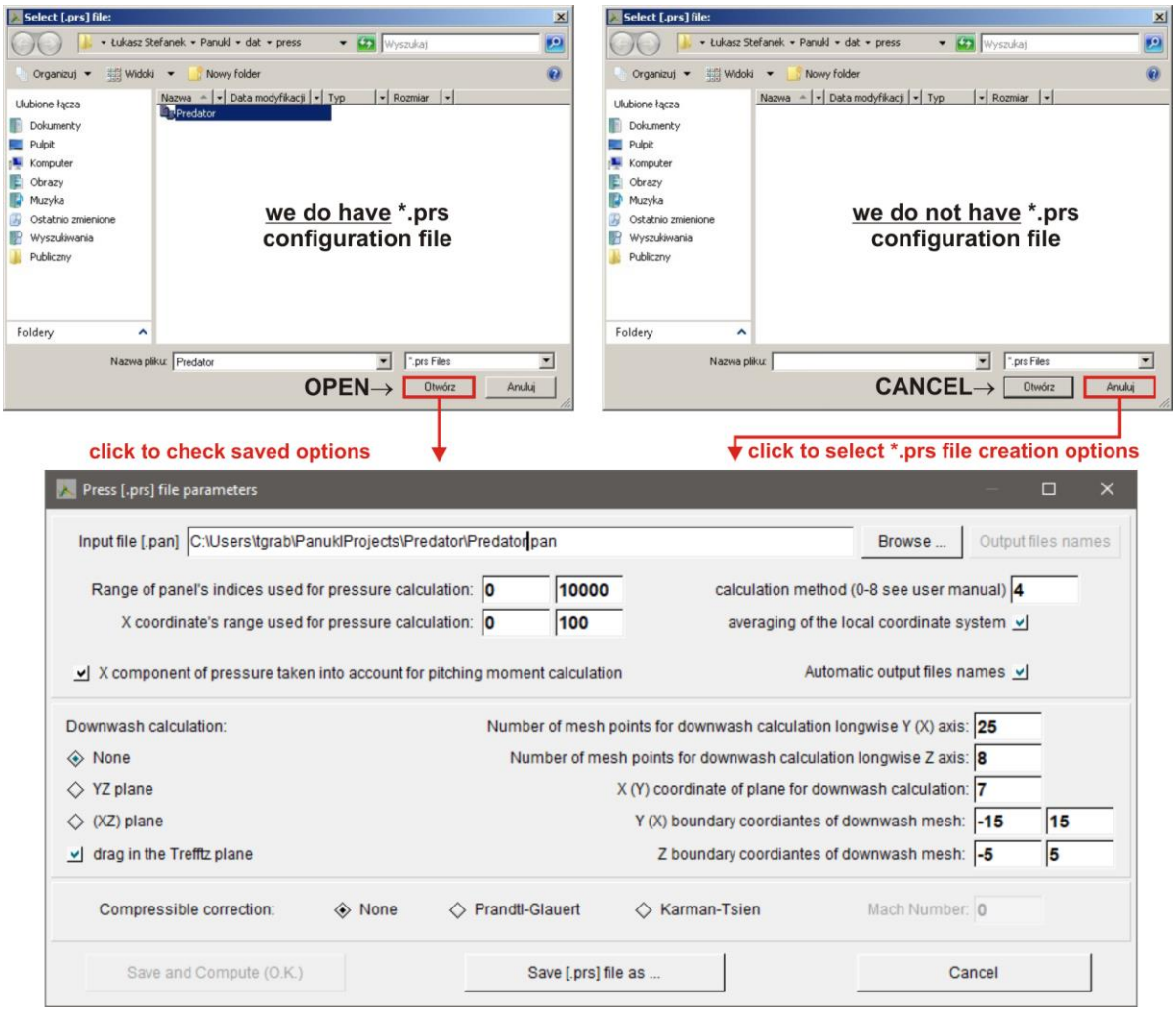

Fig. 17 – Dialog window to run program Press

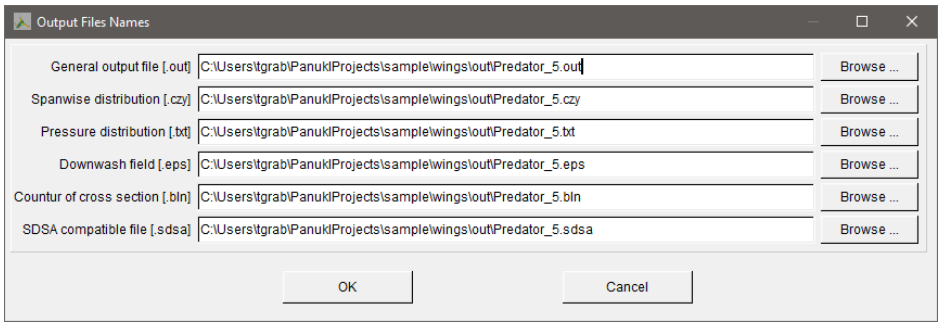

Fig. 18 – Dialog window to define output files names

Setting/Function	Description
Range of panel's indices used for pressure calculation	The values are the numbers of the first and the last panel which will be taken into account to compute global aerodynamic coefficients.
X coordinate's range used for pressure calculation	The starting and ending value of the X coordinate of the interval that will be taken into account to compute the global aerodynamic coefficients.
X component of pressure taken into account for pitching moment calculation	Selecting the checkbox will take into account the X-component of the pressure when calculating the global values of the pitching moment.
Calculation method (0-8)	Selection of the method of differentiation of potential distribution (There are four basic methods and their average ones. The basic methods are based on the determination of a polynomial based on known potential values on a given panel and his 8 neighbours): **0** – average from two out four of described below methods, **1** – collocation method – with polynomial: $\varphi(x,y)=Ax^2y^2+Bx^2y+Cxy^2+Dxy+Ex^2+Fy^2+Gx+Hy+I$, **2** – collocation method (omitting point on current panel) – with polynomial: $\varphi(x,y)=Bx^2y+Cxy^2+Dxy+Ex^2+Fy^2+Gx+Hy+I$, **3** – approximation with polynomial: $\varphi(x,y)=Bx^2y+Cxy^2+Dxy+Ex^2+Fy^2+Gx+Hy+I$, **4** – (default) approximation with polynomial: $\varphi(x,y)=Dxy+Ex^2+Fy^2+Gx+Hy+I$, **5** – method 1, 2 i 3, **6** – method 1, 2 i 4, **7** – method 1, 3 i 4, **8** – method 2, 3 i 4.
Averaging of local coordinate system	Elimination of possible errors during the potential differentiation resulting from the unfavourable "numerically" position associated with the local panel coordinate system .
Downwash calculation:	Downwash (angle of deviation) computations: **None** – downwash is not computed (result **[.eps]** file is not created), **YZ plane** – downwash results are computed in **OYZ** plane**,** **XZ plane** – downwash results are computed in **OXZ** plane
Number of mesh points for downwash calculation longwise Y (X) axis:	The number of points along the **Y** (or **X** – if **XZ plane**) axis in which the downwash angle will be calculated.
Number of mesh points for downwash calculation longwise Z axis:	The number of points along the **Z** axis in which the downwash angle will be calculated.
X (Y) coordinate of plane for downwash calculation:	The value of the **X** coordinate (or **Y** - if **XZ plane**) of the plane, in which the downwash angle will be calculated
Y (X) boundary coordinates of downwash mesh:	The initial and final value of the **Y** coordinate (or **X** - if **XZ plane**), limiting the rectangular area in which the downwash angle will be calculated.
Z boundary coordinates of downwash mesh:	The initial and final value of the **Z** coordinate, limiting the rectangular area in which the downwash angle will be calculated.

Drag In the Trefz plane	Computing the induced drag in the **Trefz** plane.
Compressible correction:	Compressibility correction method for set **Mach** number: **None** – no correction, **Prandtl-Glauert** – correction method **Karmana-Tsien** – correction method.
Automatic output filenames	Automatic result file names - result files have the same base name as [.pan] file and only extensions are changed
Output files names	Displays a dialog window to define the individual output file names – Fig. 18

Compute sequence (Shift-F10)

The command starts the PanAuto program configuration window (Fig. 19), which is used to automate the basic calculation of the package. The program allows to run one sequence of calculations, i.e. to call successively the following programs: Mesh3, Neigh, Panukl and Press with the selected configuration files. The user can observe the status of the calculation. The color of the round widget (Status) indicates the operating status: no color - the process has not yet been started, red - during the calculation, green - completed calculation.

Program can also run in the batch mode, using the following command:

PanAuto [filename.auto] [iGUI]

where:

filename.auto – configuration file (see [6])
iGUI – GUI flag (0 –off, 1 - on) (Fig. 19)

The program without parameters is always run with the graphical interface (Fig. 19). Entering only the first parameter starts the program without graphical interface.

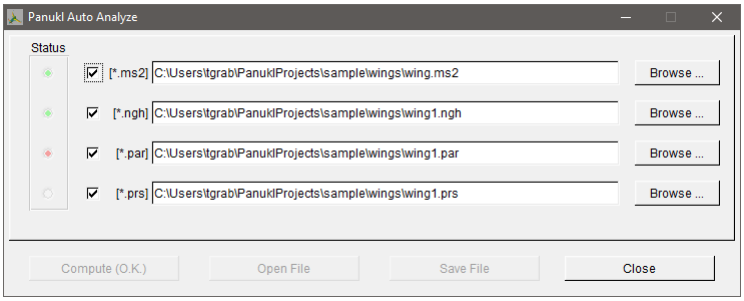

Fig. 19 – Dialog window of PanAuto program

Prepare data for SDSA

The command runs the **PanProject** configuration window (Fig. 20), which is used to calculate complete aerodynamic data, including aerodynamic derivatives, in a form compatible with the SDSA package [7]. The set of data (parameters) of the **PanProject** program is set using the dialog window widgets and can be saved in a file type [.prj]. The meaning of individual variables is presented in a separate document [6]. The user must prepare the file in advance with the object's grid and the

vortex wake [.dat]. To compute control derivatives, it is necessary to prepare a grid version with deflected control surfaces (elevator, ailerons, rudder). Because the process of calculating the data set to the SDSA can be time-consuming, the user can set that the calculations take place in parallel, using the multicore/multiprocessor architecture of the computer. It is recommended that the number of parallel processes be smaller than the number of cores/processors installed in the computer.

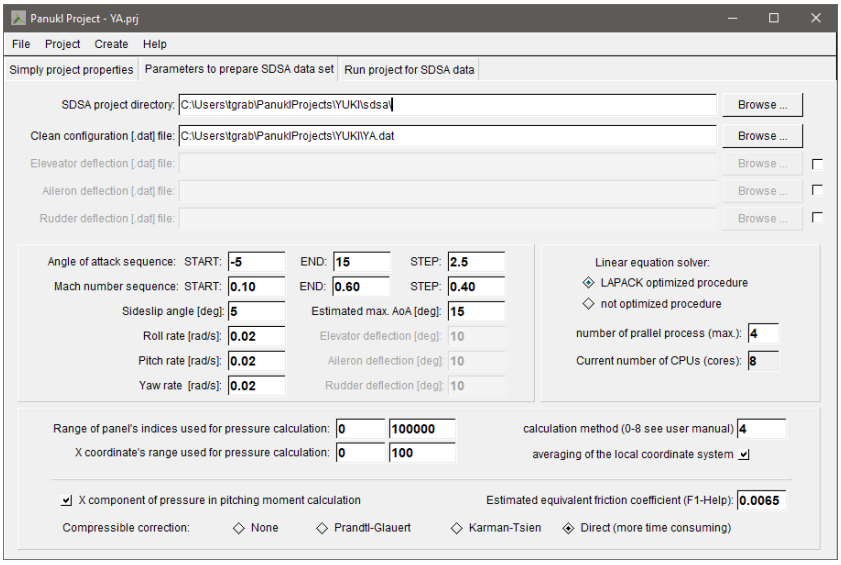

Fig. 20 – Dialog window of the PanProject program in the preparing data to SDSA mode

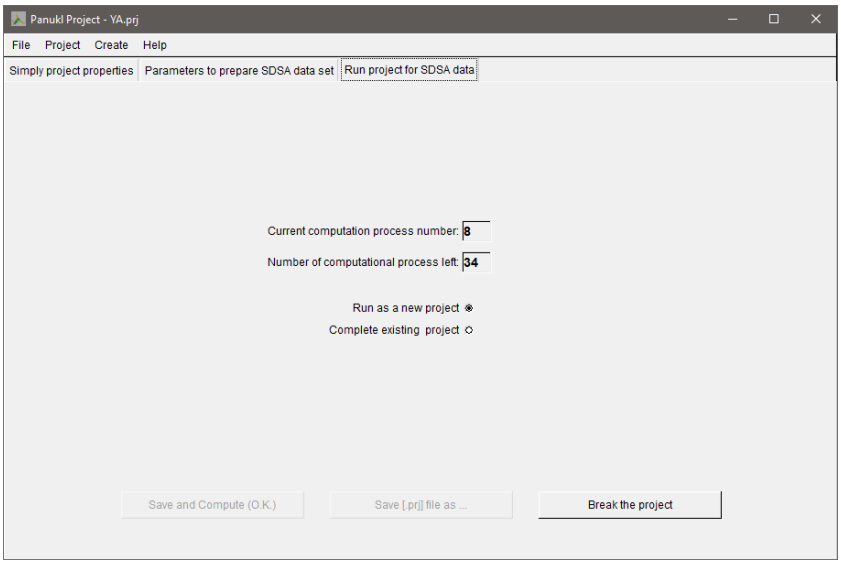

Fig. 21 – Dialog window of the PanProject program in project run mode for SDSA

After setting all the parameters, go to the "Run project for SDSA data" tab (Fig. 21) and begin calculations with the "Save and compute (OK)" button. The project parameters are saved to the [.prj] file and the calculation procedure begins. The calculation progress is shown on two counter widgets. The first one shows the currently running last process (sequence of Panukl and Press programs), the second shows the number of sequences that remains to be performed. The "Break the project" button is used to stop calculations. The calculations can then be resumed at any time, even after the PanProject program is called again.

The PanProject program can be also used to compute a single sequence, similary as the PanAuto program. Calling a single sequence is possible after switching the program to the "Simply project properties" tab (Fig. 22). The "Create" menu options enable manual start of individual calculation steps, similar to the GridView program options. Dialog windows are available in the tabs Fig. 23.

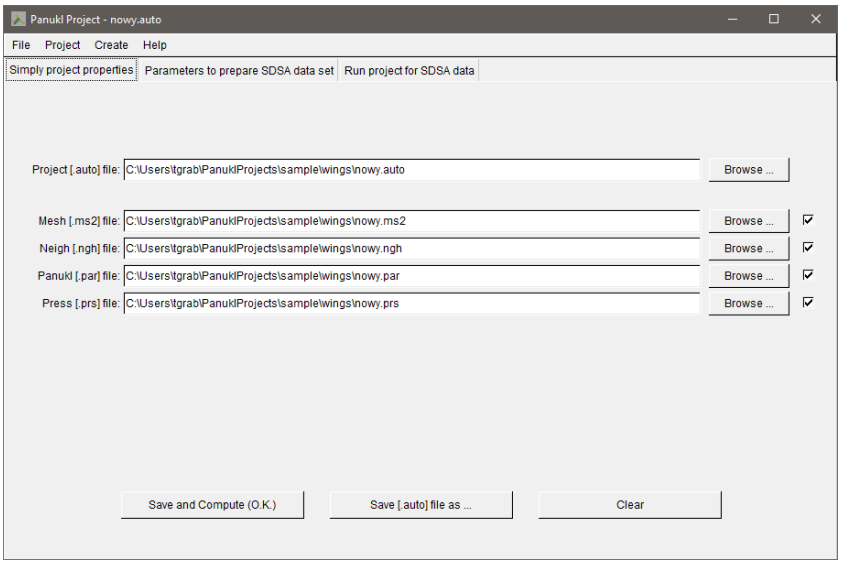

Fig. 22 – Dialog window of PanProject program in one calculation sequence mode.

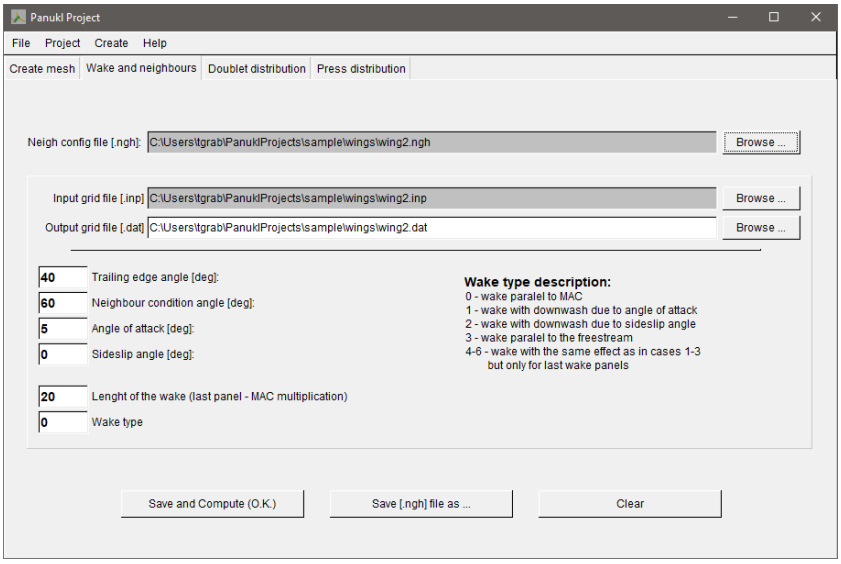

Fig. 23 – Dialog window of one of the options (Wake and neighbours) of Create menu

Program can also run in the batch mode, using the following command:

PanProject [filename.auto/filename.prj] [iBatch]

where:

filename.auto/filename.prj – configuration file (see [6])

iBatch – batch mode flag (0/1).

Depending on the type of the input file, the program will run in simple sequence mode (file type [.auto]) or in the mode of preparing data for SDSA (file type [.prj]).

Export FEM data for Calculix

The command runs the configuration window of the **FEM** program (Fig. 24), which is used to export data and calculation results (pressure distributions) to the Calculix [4] program, which carries out strength analyzes using the finite element method. A detailed description of the operation and the manual of the program can be found in a separate document [5].

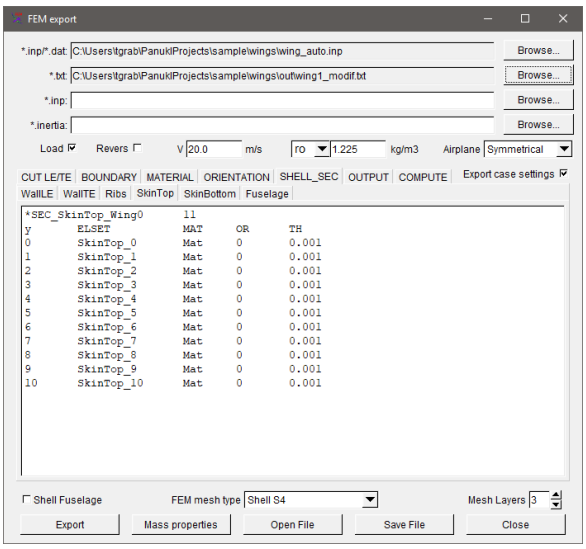

Fig. 24 – Dialog window of FEM program

Connect two grids

The command runs dialog window to configure and call Connect program (Fig. 25), which is used to connect two grids with vortex wakes [.dat].

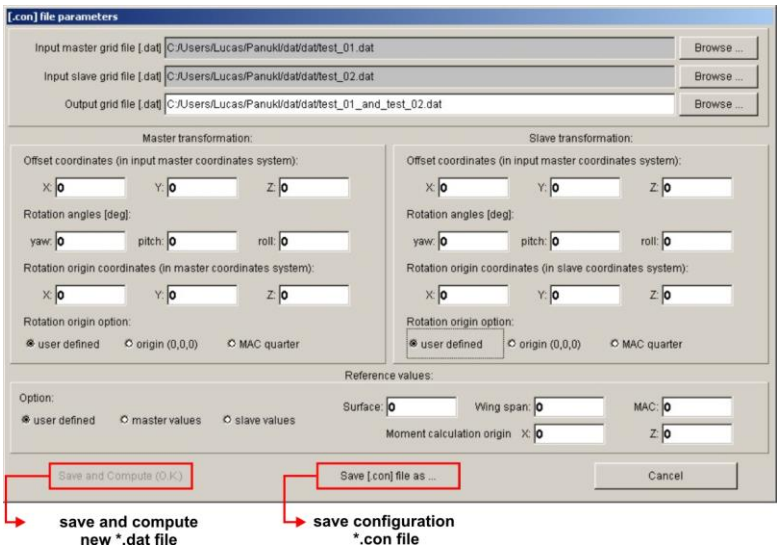

Fig. 25 – Dialog window to configure the procedure of connection two grids

Thanks to this option, it is possible to create the complex grids consisting of more than one object and create unsymmetrical, complicated meshes. This option gives the user of the **PANUKL** package the possibility of, for example, analysis of the mutual aerodynamic impact of one object to

another (when two or more objects are in close neighbour - Fig. 26) or disconnection analysis, e.g. a fuel tank dropped from an airplane, etc.

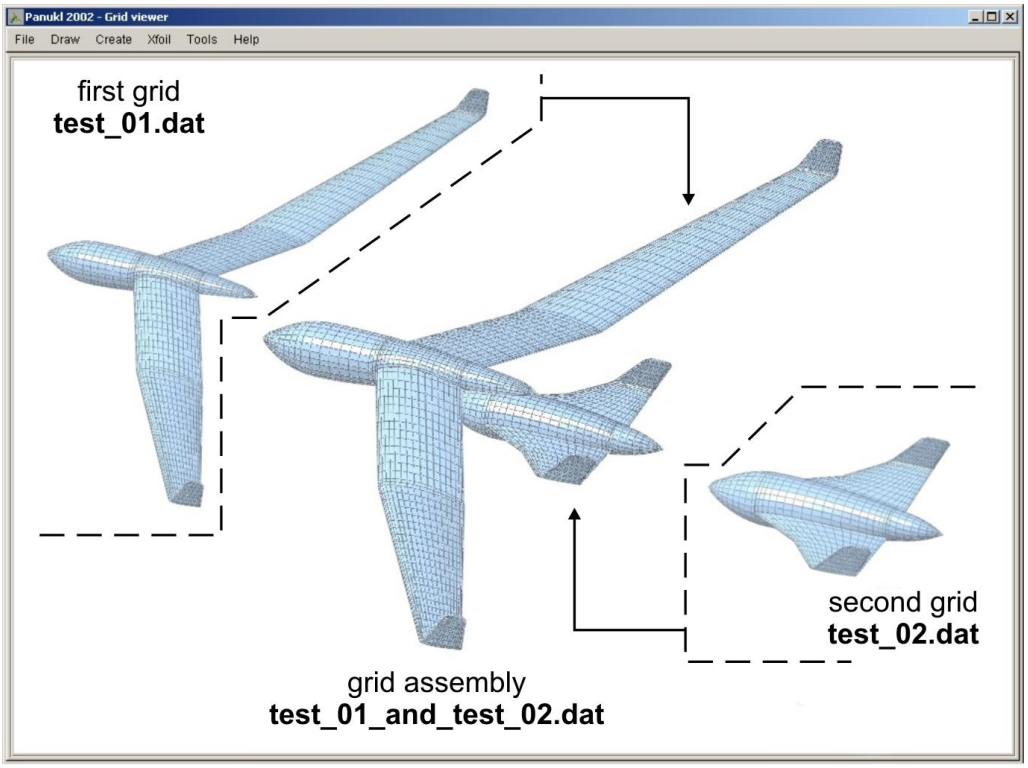

Fig. 26 – What does CONNECT TWO GRIDS function do ?

How it Works ?

Option 1 – we do have configuration **[.con]** file (Fig. 27)

Run **CONNECT TWO GRIDS** and select saved configuration file **[.con]** – file contains all necessary information to create **[.dat]** file (which will be an assembly of two existing grids). To open selected **[.con]** file click **OPEN** button. Configuration window will appear (Fig. 25) where one can see saved **[.dat]** file creation options. To generate **[.dat]** file click **Save and Compute (OK)** button.

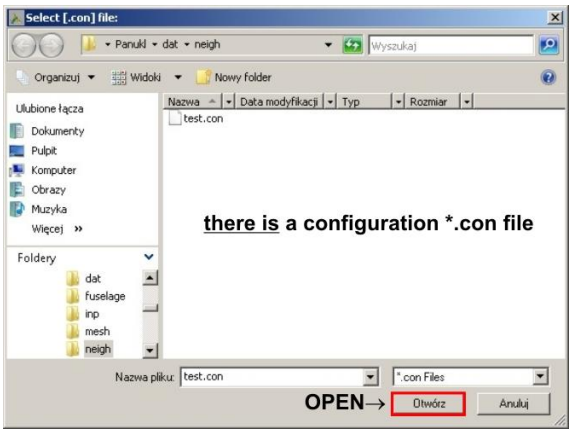

Fig. 27 – Open the connection configuration file [.con]

Option 2 – we do not have configuration **[.con]** file

Run **CONNECT TWO GRIDS** and click **CANCEL** button when prompted for saved configuration file **[.con]**. The configuration window will appear (Fig. 28) where user can select options to create **[.dat]** file. To save current **[.dat]** creation options to **[.con]** file click **Save [.con] file as**, to create **[.dat]** file click **Save and Compute (OK)** button.

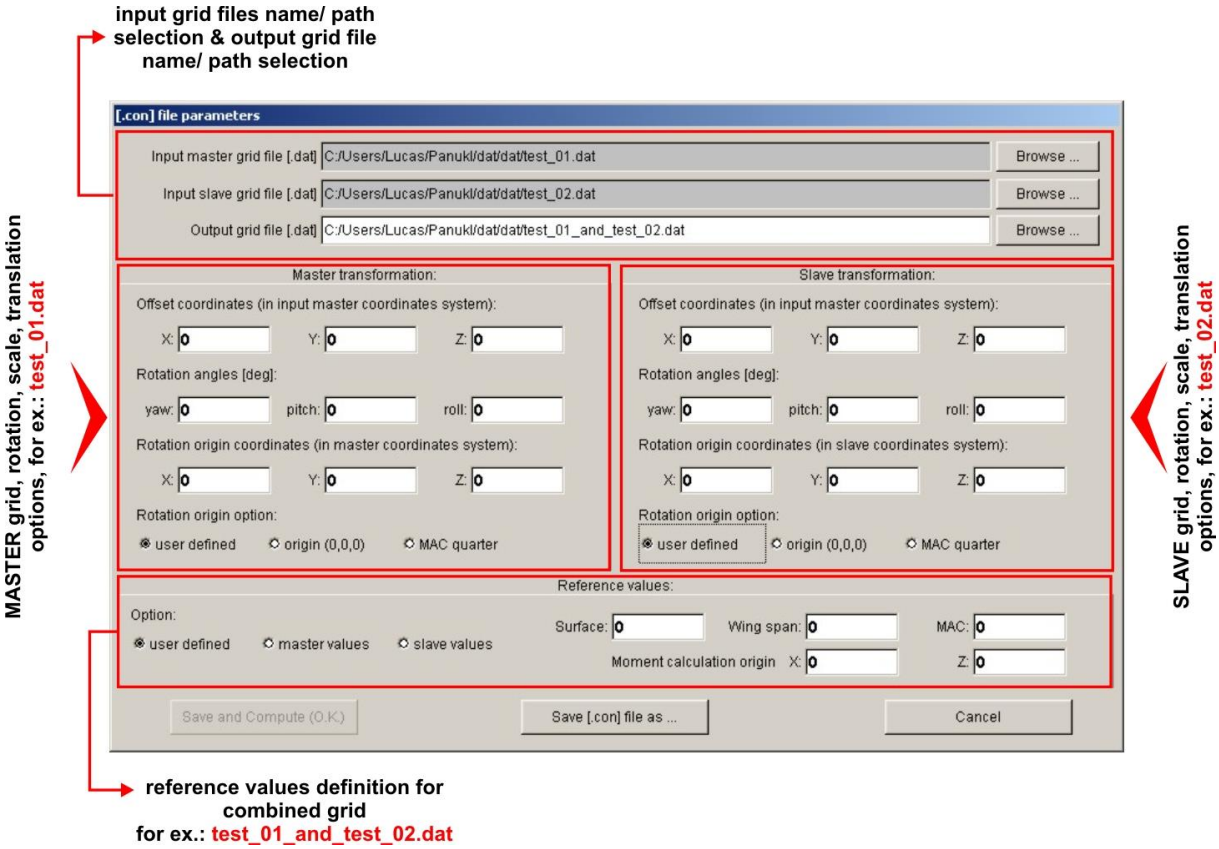

Fig. 28 – CONNECT TWO GRIDS main options

Function	Description
Offset coordinates (in input master coordinates system)	Offset coordinates (in input master coordinates system) – **X, Y, Z**
Rotation angles [deg]	To rotate component enter the necessary rotation angles [deg]. Rotation origin can be defined as: **User defined –** defined by user, **Origin (0,0,0) –** the origin of coordinate system for current object, **MAC quarter –** ¼ MAC for current aircraft.
Reference values	User must specify reference values for target output object: **User defined** – defined by user, **Master values** – reference values will be taken from master grid model **Slave values** – reference values will be taken from slave grid model

Correct neighbours

The command runs the dialog box (Fig. 29) to start the **Neigh_con** program, which is used to correct "neighbours" for the already generated data file [.dat]. The neighbours correction program must be run if the grid connection procedure described in the previous chapter has been used.

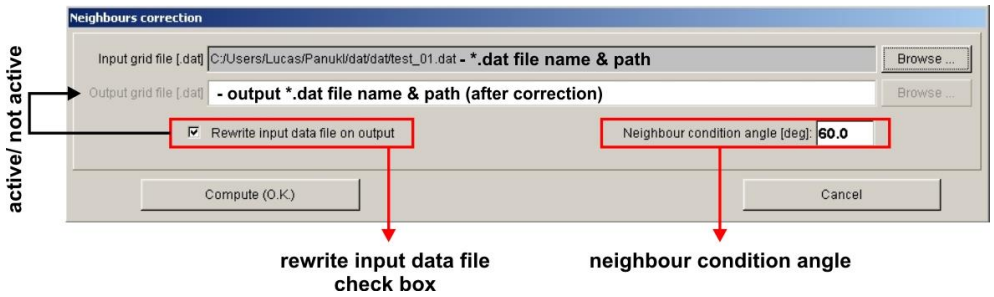

Fig. 29 – Dialog window of “Correct Neighbours” option

Correct geometry

The command runs the dialog box (Fig. 30) to start the Geom_cor program, which is used to correct the panel geometry in the connection plane when creating a combined mesh of several components. The user must select the grid file with the neighbours (Input grid file) and the file to which the result grid is to be saved (Output grid file). In addition, it should be specified in which plane correction is to be made by specifying the type (three possibilities: X - YZ plane, Y - XZ plane, Z - XY plane) and the exact coordinate value. The position of nodes located in this plane is adjusted so as to obtain the consistency of the grid.

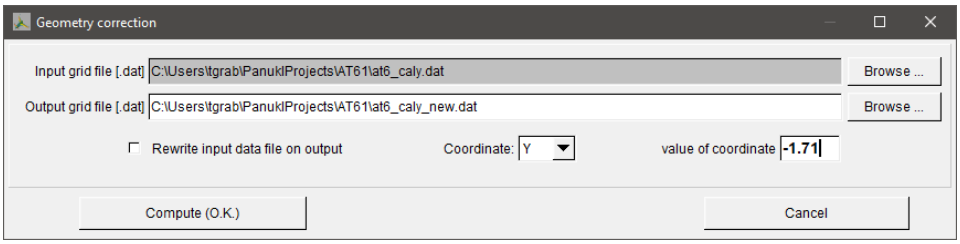

Fig. 30 – Dialog window of „Correct geometry” option

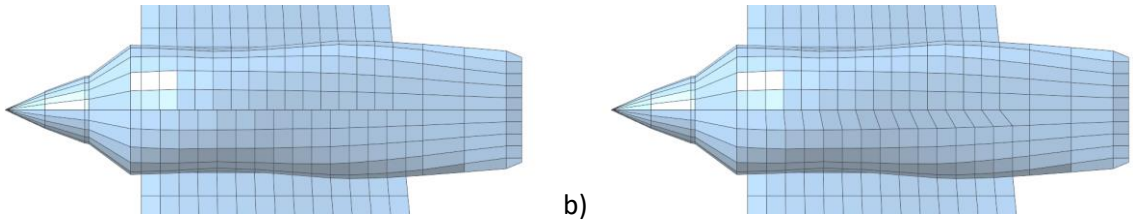

Fig. 31 – Geometry correction in the central plane of engine nacelle: a – before correction, b – after correction

Note: The "Correct geometry" option is considered at the present stage as experimental and should be used with caution due to the possibility of "holes" in the surface of the plane (open panels).

XFOIL menu description

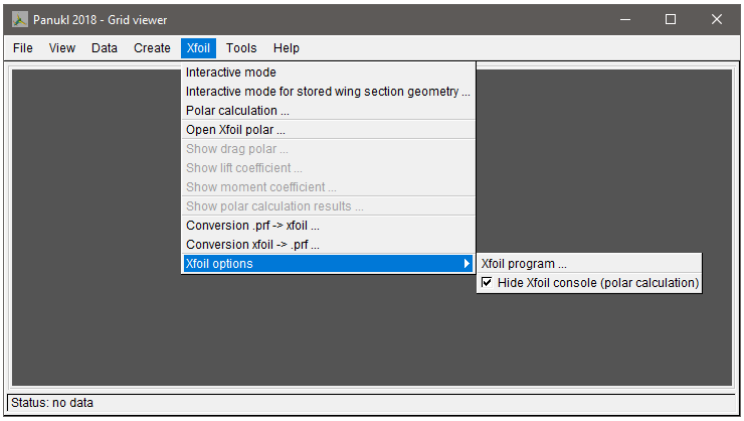

Fig. 32 – Menu – XFOIL

In the **XFOIL** menu, the user will find the most important functions to facilitate the aerodynamic analysis of the profiles using an external **XFOIL** program. In order to correctly use and interpret the obtained results, a basic knowledge of the **XFOIL** program is required..

Calculation options

Function	Description
Interactive mode	Click to run external **XFOIL** program. Standard program window will appear
Interactive mode for stored wing section geometry	Run the external XFOIL program for the selected file **[.dat]** containing a profile geometry data that will be subjected to aerodynamic analysis (Fig. 34).
Polar calculation	Computing the basic aerodynamic characteristics for an airfoil: **CL-lift**, **CD-drag**, **CM-moment**, versus **angle of attack** and **Reynolds** & **Mach** number. (airfoil geometry saved to **[.dat]** file). Results of calculation of the aerodynamic characteristics are saved to [**.txt]** file (Fig. 33).

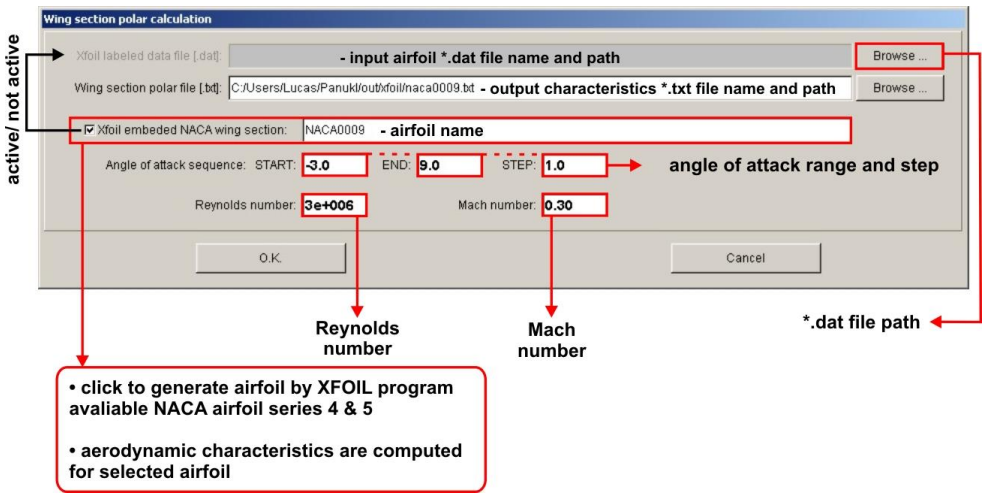

Fig. 33 – Polar calculations setup window

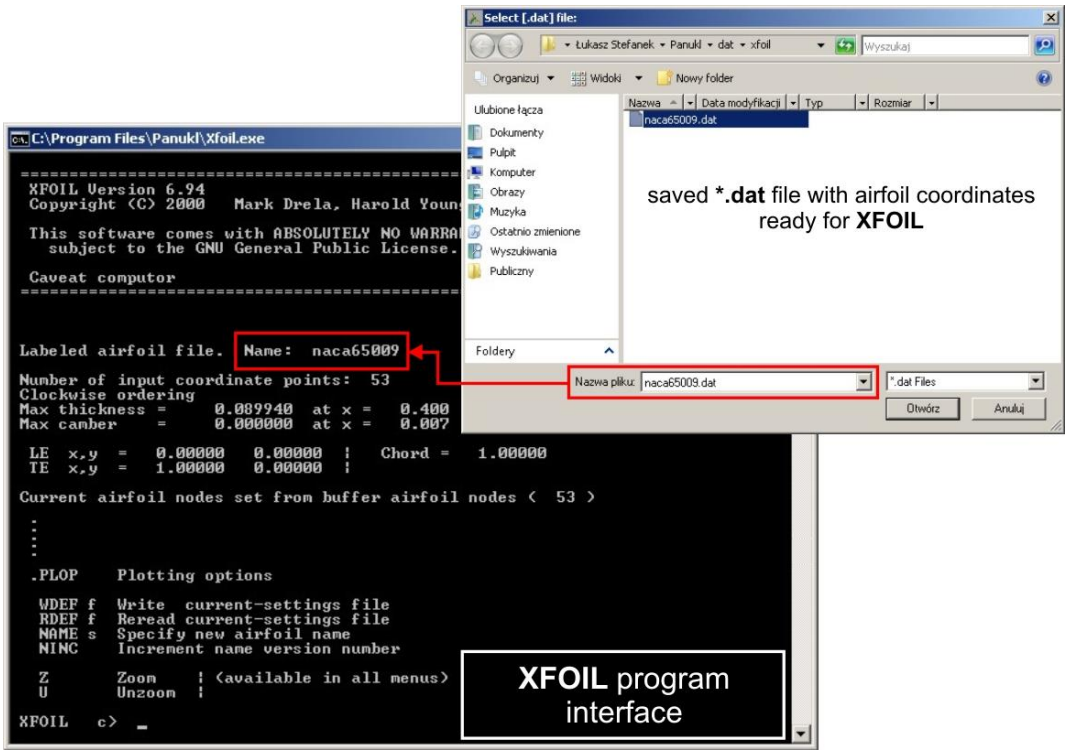

Fig. 34 – External XFOIL program window and airfoil [.dat] file selection window

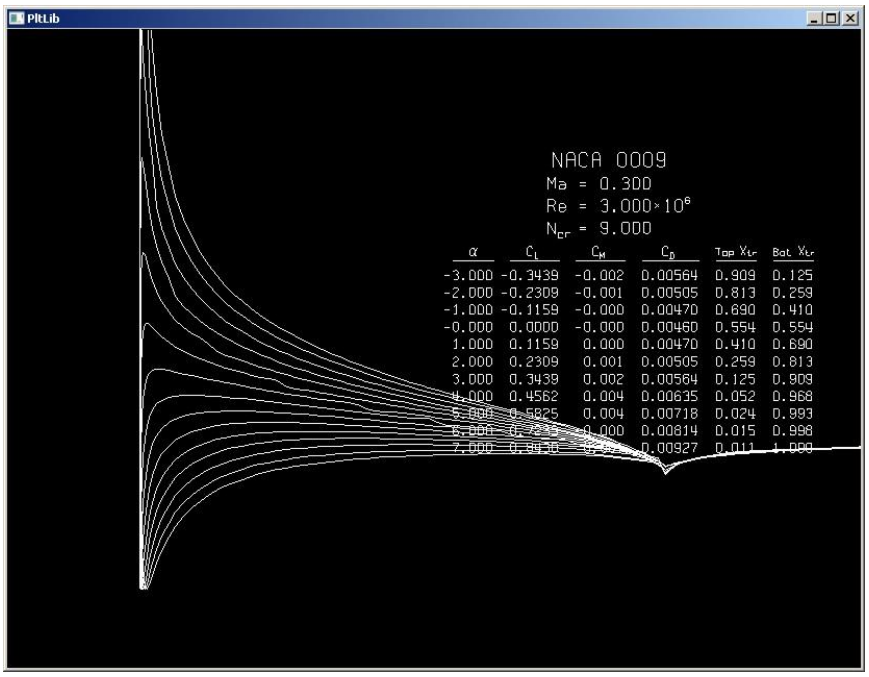

Fig. 35 – XFOIL window – aerodynamic coefficients computations for an airfoil

During **XFOIL** aerodynamic computations user must check if results converge.
Otherwise obtained results can have no physical sense. For more information go to **XFOIL** manual.

Preview options for results

Funkcja/ Polecenie	Opis
Open XFOIL polar	Loading the file with saved aerodynamic characteristics of the profile from the selected location on the disk, Fig. 36. After loading the data file [.txt], the following commands become active:
Show Drag polar	Displays a window with drag characteristic **CD** , Fig. 37.
Show Lift coefficient	Displays a window with lift characteristic **CL**, Fig. 37.
Show Moment coefficient	Displays a window with moment characteristic **CM**, Fig. 37.

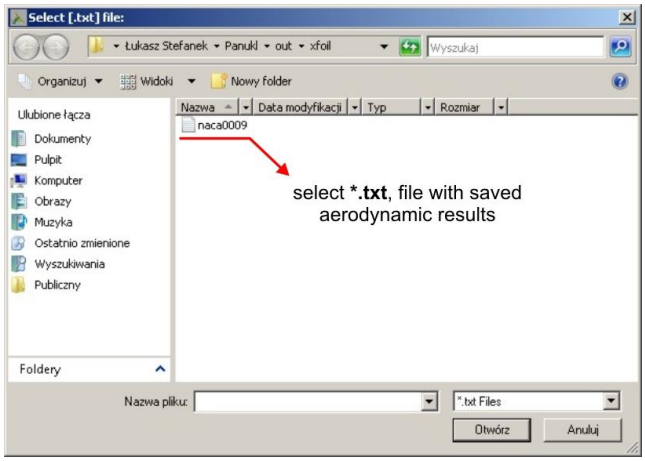

Fig. 36 – Selecting result file with aerodynamic characteristics for an airfoil

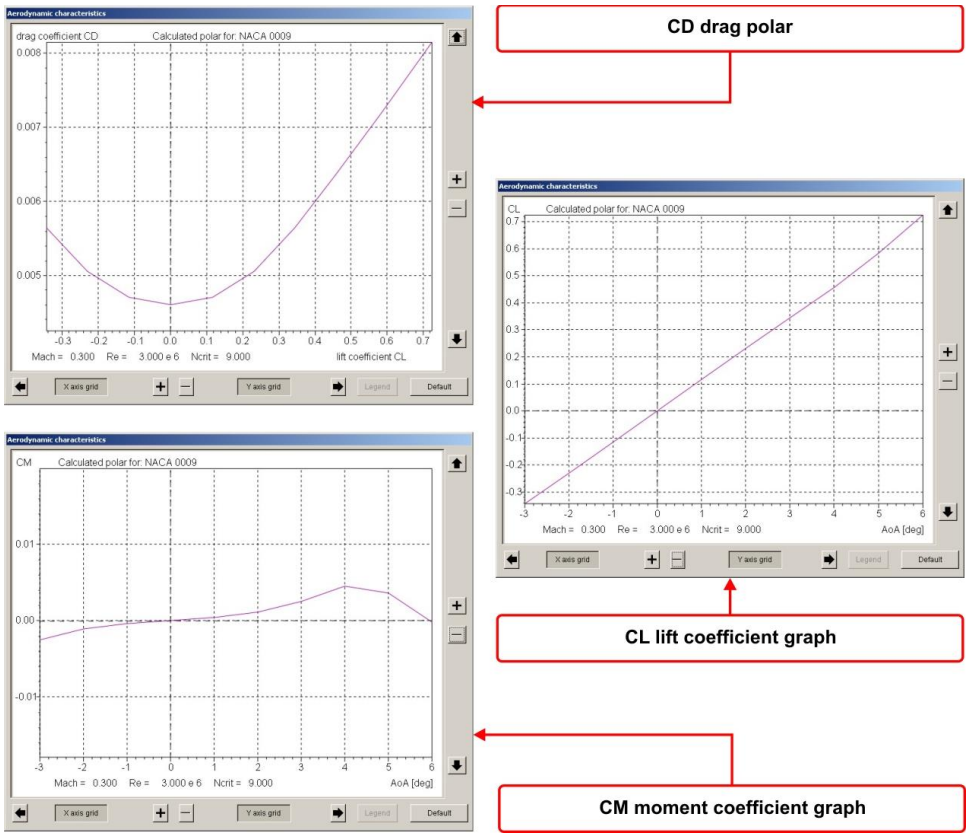

Fig. 37 – Example results CL, CD, CM versus Angle of Attack

Tool options

Funkcja/ Polecenie	Opis
Conversion [.prf] to XFOIL	Option to convert a [.prf] file containing a profile geometry definition in the native format for the PANUKL program to [.dat] format accepted by XFOIL, Fig. 38.
Conversion XFOIL to [.prf	Option to convert a [.dat] file containing a profile geometry definition in a format accepted by the XFOIL program to the [.prf] format native to the PANUKL program, Fig. 38.
Xfoil options	The option contains two additional program settings XFOIL: • Xfoil program – sets the location of the Xfoil executable file; the default is to set the program distributed with the package, however, the user can use the program installed separately or located in the operating system distribution (Linux). • Hide Xfoil console (polar calculation) – the Xfoil program, which is running for the polar calculation of the profile, opens the window of the console on which it shows the course of calculations; this option hides the console window and calculations take place in the background.

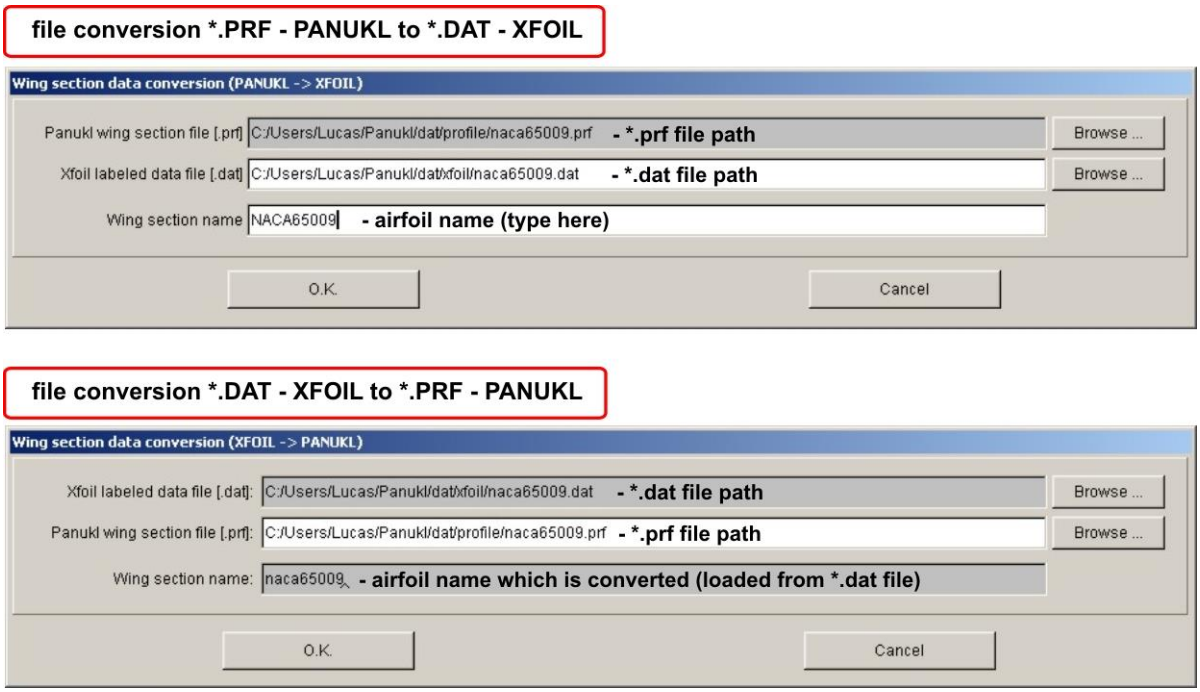

Fig. 38 – Airfoil file type conversion

TOOLS menu description

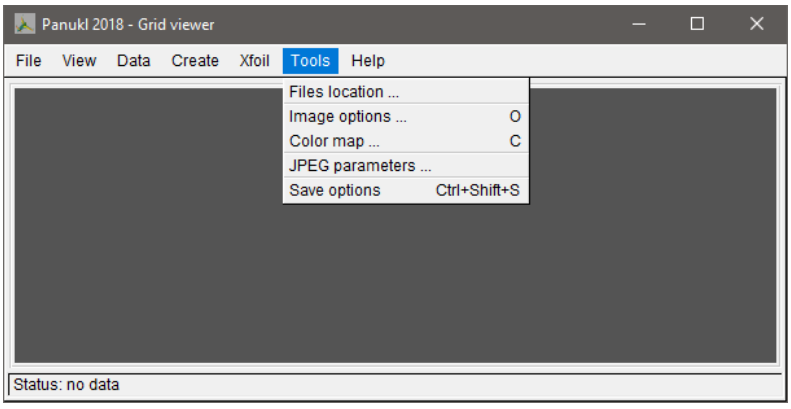

Fig. 39 – TOOLS

Files location

This option displays a dialog box in which the user can change the settings regarding the location and names of the folders in which the input data is stored, the results obtained and other files related to the operation of the package (Fig. 40).

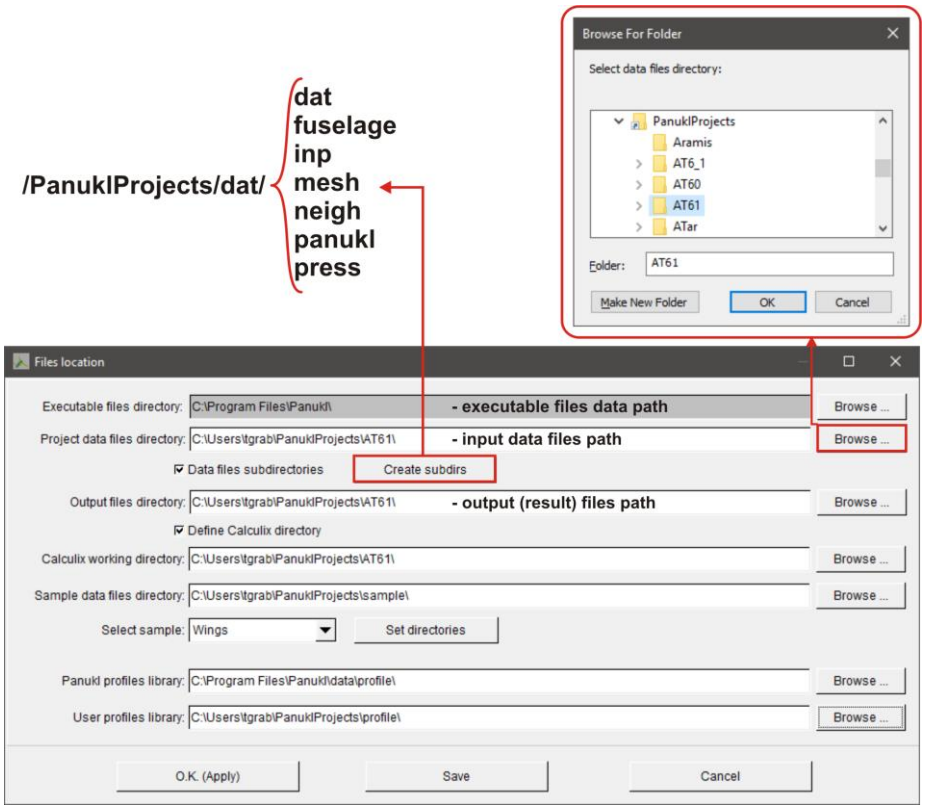

Fig. 40 – Files location selection window

The user must set up the directories for the data files (Project data files directory) and for the output files (Output files directory). It can be the same directory (project directory) or different specialized catalogs. The user can choose specialized subdirectories for data files:

- script files to mesh generation – subdirectory **mesh**
- fuselage geometry files – subdirectory **fuselage**
- files containing the mesh created by Mesh3 program – subdirectory **inp**
- configuration files for Neigh program [.ngh] – subdirectory **neigh**
- files containing the mesh with the vortex wake – subdirectory **dat**
- configuration files for Panukl program [.par] – subdirectory **panukl**
- configuration files for Press program [.prs] – subdirectory **press**

Click **CREATE SUBDIRS** button to automatically create proper directory structure for **PANUKL**'s input and output data. . Click **SAVE** button to save options

Image options

The option displays a dialog box with options on how to display the model in the graphics window (Fig. 41).

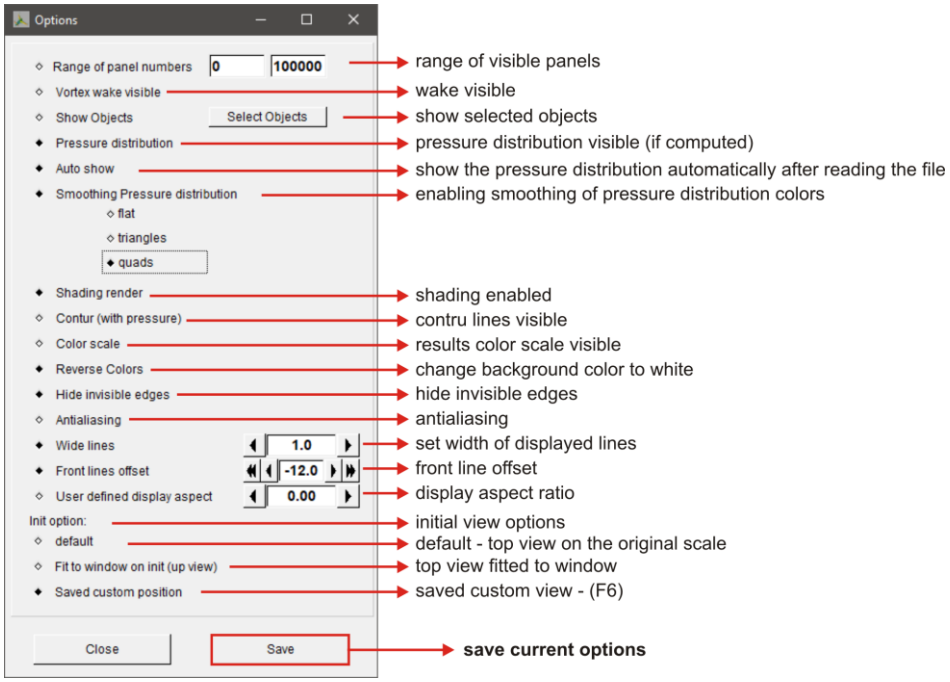

Fig. 41 – The window for selecting options regarding the display of the model

Color map

The option displays a dialog window in which the user can change and set the way in which the results of distributions, eg pressure, speed, etc., will be presented (Fig. 42).

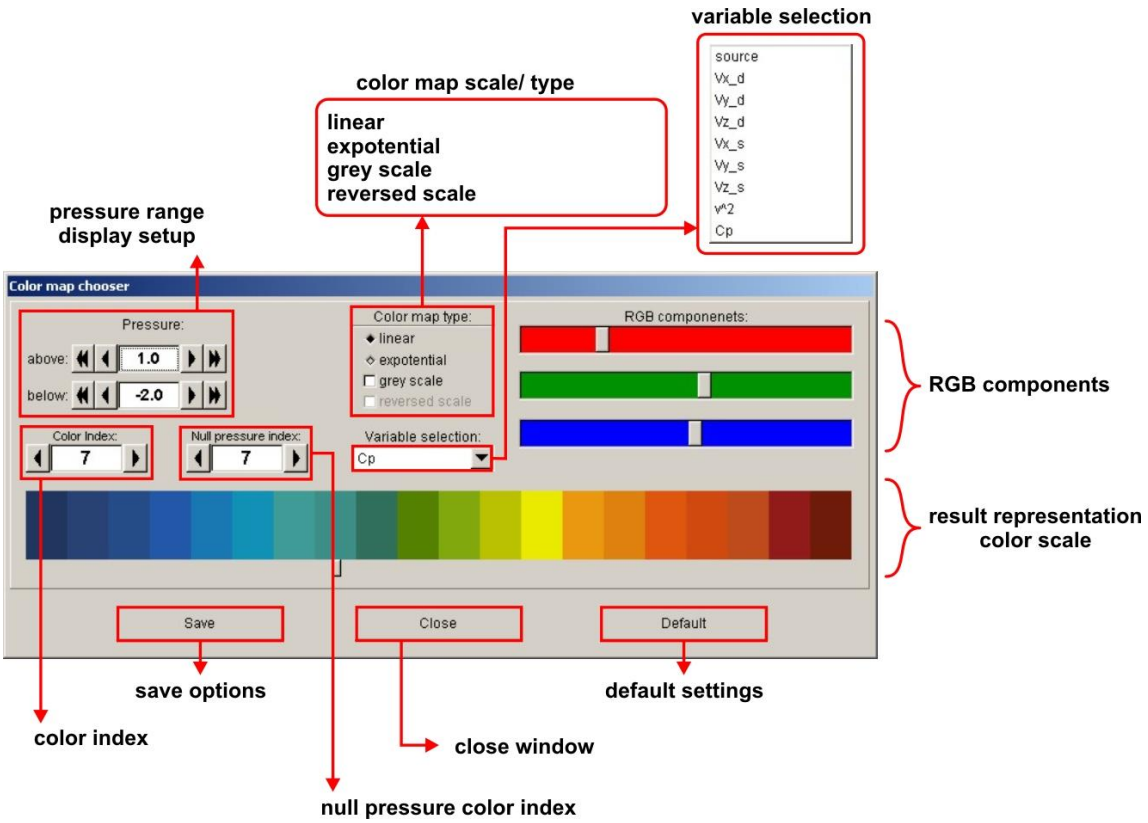

Fig. 42 – Graphic representation of results - setup window

JPEG parameters

The option displays the window where user can adjust **JPEG** picture capture quality and options (Fig. 43).

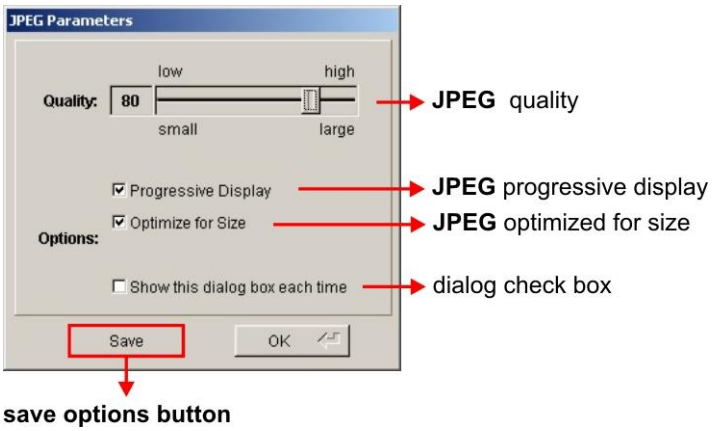

Fig. 43 – JPEG screen capture setup window

Save Options (Shift+Ctrl+S)

Click to save current options of PANUKL package.

HELP menu description

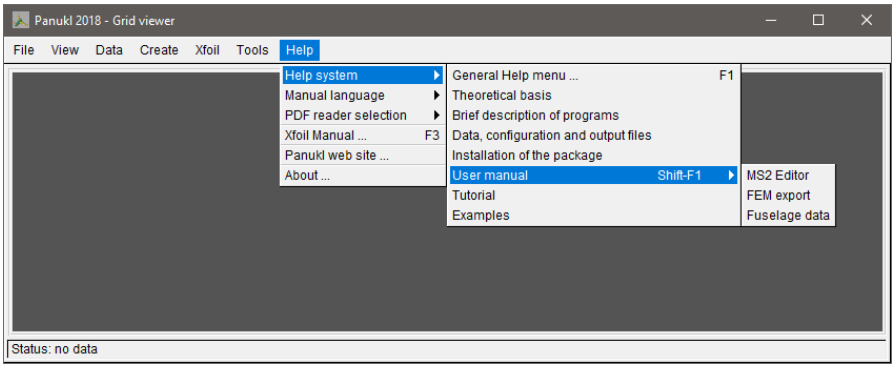

Fig. 44 – Menu – HELP

HELP menu – available options

Function	Description
Help system (F1)	General Help menu
Manual language	Help language (available: **PL** i **ENG**)
PDF reader selection	PDF document reader application selection.
Xfoil Manual (F3)	XFOIL manual [3]
Panukl web site	PANUKL home webpage
About	Click to display about **PANUKL** window, Fig. 45.

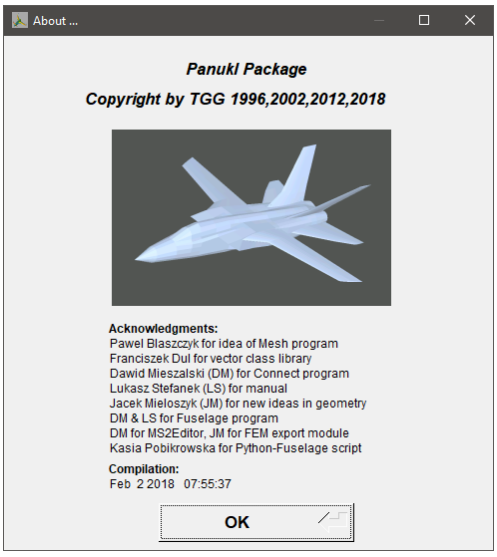

Fig. 45 – Information window

Help system

The help system includes a main help file that contains general information about the package and links to individual documentation files. The entire documentation includes theoretical foundations, a description of basic functionalities and methods of use as well as examples of application. The help system includes:

- General Help menu (F1)
- Theoretical basis
- Brief description of the package components (programs)
- Data, configuration and output files
- Installation of the package
- User manual (Shift F1 – this document)
    - MS2 editor –MS2 editor description
    - FEM export – export of data and results to Calculix [4]
    - Fuselage data – description of program to import fuselage data from NX
- Tutorial
- Examples – description of the samples included in the package

References

[1] MS2 editor description
[2] Fuselage program description
[3] Drela M., (2017) Xfoil - Subsonic Airfoil Development System, http://web.mit.edu/drela/Public/web/xfoil/
[4] CALCULIX - A Free Software Three-Dimensional Structural Finite Element Program, http://www.calculix.de/
[5] FEMexport program description
[6] Data, configuration and output files
[7] Goetzendorf-Grabowski T., (2018), Simulation and Dynamic Stability Analysis, https://www.meil.pw.edu.pl/add/ADD/Teaching/Software/SDSA