universal mechanism

Getting started

UM Durability

Getting started

2024

Contents

Getting Started Using UM: UM Durability Module

This manual leads you through an example of using UM Durability module. It is supposed that you already have studied the gs_UM.pdf[1] manual, which is devoted to basics of UM modeling and know how to create a new model, add new bodies and joints, generate and compile equations of motion (UM Input) and simulate mechanical systems (**UM Simulation**), as well as gs_UM_FEM.pdf[2], which is devoted to introducing flexible bodies into UM models.

In this manual we will consider all steps of an example of stress loading and durability analysis in Universal Mechanism. We will run some simulations of dynamics of this system and then having results of these simulations will estimate its stress loading and life time.

The example of analysis of stress loading and durability properties of the vibration table illustrates the general sequence of operations with the help of UM Durability module. In this example the common S-N method for durability prediction is used.

Compatibility

Before coming to the rest part of the manual please check if **UM Durability** is available on your computer. Run **UM Input** or **UM Simulation** and from the **Help** menu select **About…** The list of available modules is shown in the **Configuration** section.

Copyright and trademarks

[1] www.universalmechanism.com/download/90/eng/gs_um.pdf
[2] www.universalmechanism.com/download/90/eng/gs_um_fem.pdf

1. Introduction

The present paper describes the CAE-based durability analysis that is being implemented in Universal Mechanism software to predict the fatigue damage of parts of mechanical systems. The dynamic simulation of parts is performed using Universal Mechanism. Flexible body representation is considered using external finite element (FE) software. The present version of UM supports two FE software – ANSYS and MSC.NASTRAN. The final durability post-processing analysis is performed using the UM Durability module.

The analysis starts with the dynamical hybrid model in Universal Mechanism. The flexibility characteristics of the structural parts are incorporated into UM model using a modal formulation based upon component mode synthesis. Basically, this method represents the part's flexibility using a modal basis, which is optimized to account for constraint and force locations. The mode shape displacements and stresses are calculated using the finite element programs ANSYS or MSC.NASTRAN.

The UM Durability module combines the stress time history information generated during series of numerical experiments in UM and the material fatigue strength characteristics to generate the predicted life distribution in the part.

Any durability analysis relies on three key inputs:

- stress loading data – time history of the stresses;
- material data – how the material reacts to repeated stress application at various stress levels;
- durability parameters calculation method.

By employing the full finite element representation of the component in the UM model, the local stresses are directly obtained as result of the UM solution. In the UM FEM module, flexible body deformations are modeled as a linear combination of mode shapes. As long as the number of mode shapes selected adequately the modal superposition will model deformations accurately and efficiently.

The idea that the deformation of a flexible body can be represented by the sum of a number of mode shapes, scaled by appropriate factors, can be extended to stresses in the body as well. These factors, or modal coordinates, can be used as the scaling factors on the stress solution of each mode shape and the superposition of these scaled stresses represents the body's instantaneous stress state. If the superposition is performed at every node in the finite element model, for every time step in the UM solution, the stress time history is defined at every location.

Using the UM FEM tools the modal coordinate time history can be saved for every numerical simulation in UM. Based on this modal time history and file with orthonormalized mode shapes from ANSYS or MSC.NASTRAN the stresses at every node can be obtained.

When UM Durability is started, the user is prompted for the location of modal coordinate time history files and orthonormalized mode shapes files, then the type of analysis required and the material data to be used.

With all of the parameters set, UM Durability performs the stress at every node, and then proceeds to multi channel peak/valley extraction and rain flow cycle counting, followed by the damage sum.

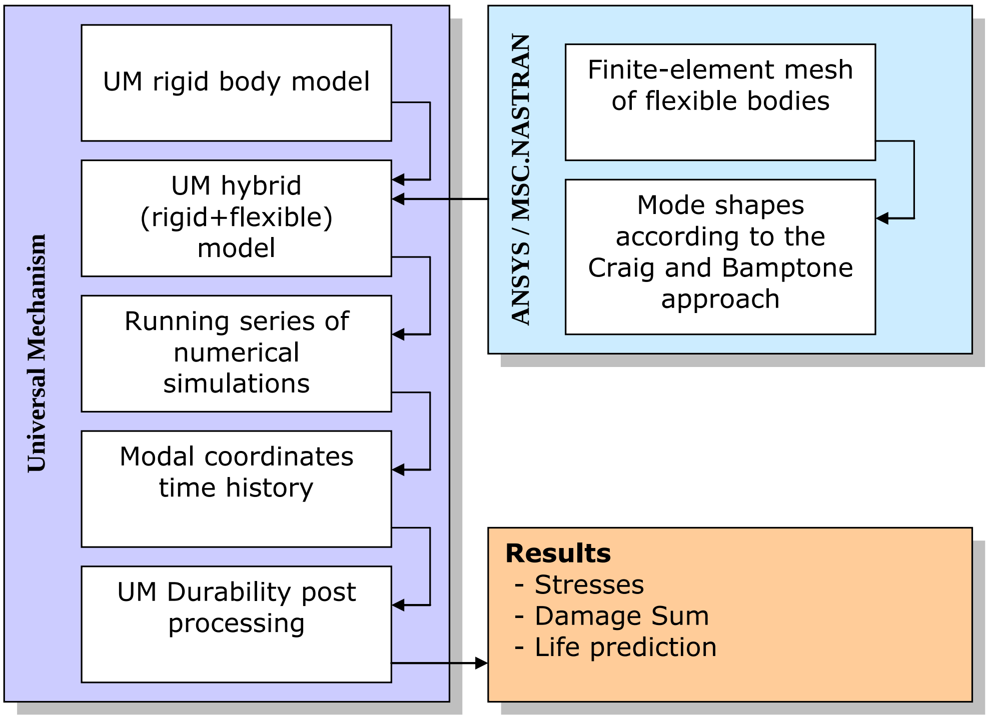


Figure 1.1. Workflow

2. Platform

Creating UM-model of the considered system is discussed in details in the Sect. 3 of gs_UM_FEM.pdf and consists of an electric motor and an elastic platform. Only questions of fatigue analysis are considered here. Please download the ready-to-use model we will work with and extract it into any directory. Use the following link to download: www.universalmechanism.com/download/90/eng/vibrostand.zip[3]. In this manual we will address the directory with this model as {UM Data}\ SAMPLES\Durability\Vibrostand.

[3] In comparison with the model described in the Sect. 3 of **gs_UM_FEM.pdf** this model has refined FE-mesh for more accurate results

2.1. Model description

The object of this research is an elastic platform, used by the support of the electric motor, see Figure 2.1. The aims of investigation are stress loading parameters calculation (such as stress values, stress amplitudes, etc.) and an estimation of fatigue strength of the elastic platform.

This FEM model of the elastic platform includes 3456 elements of SHELL63 type. Width of all elements is 5 mm. 24 static modes and 10 eigenmodes are used for the motion description.

The motor and platform suspensions consist of 8 viscous-elastic and 8 dissipative linear forces.

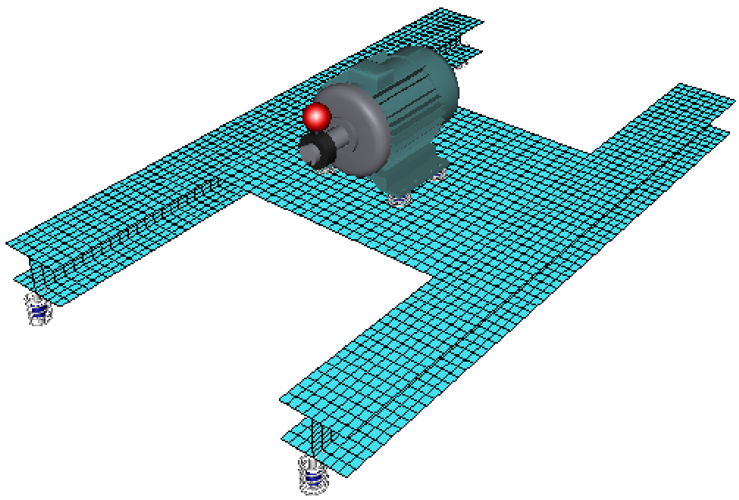


Figure 2.1. Vibration table model

2.2. Workflow

According to Figure 1.1, having the prepared UM hybrid model we need to run number of numerical experiments and then based on their results obtain estimation of damage sum and life prediction.

2.3. Dynamical analysis

To obtain results for further durability analysis we need first to perform some numerical experiments with this model.

Start **UM Simulation** program and load the **Vibrostand** model that you just extracted from the downloaded archive.

The model includes three modes: speeding up, stable work and braking operation regimes of the vibration table. Angular velocity of the rotor of the motor is changed according to the law, shown in 2. Basic model parameters are presented in Table1.

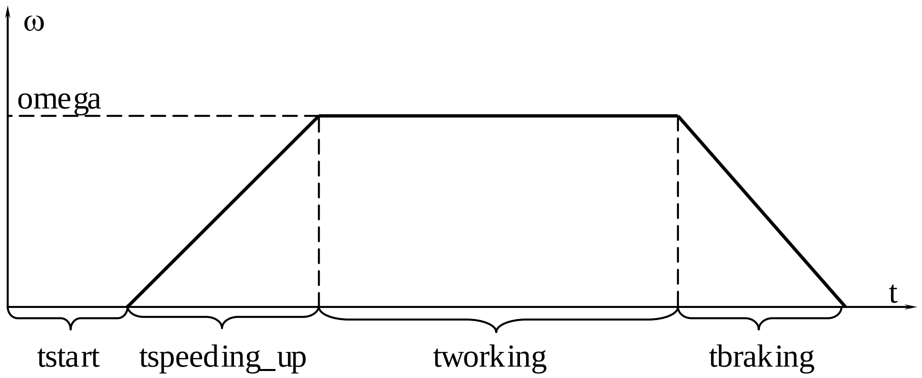


Figure 2.2. Angular velocity of the rotor

Table1

Basic model parameters

Model	Comment	Value
Nu	Nominal angular velocity of the rotor (r.p.m.)	1620
omega	Nominal angular velocity of the rotor, rad/sec	169.6
tstart	Time before speeding up, sec	0.5
tspeeding_up	Time of speeding up mode, sec	2
tworking	Time of working mode, sec	3
tbraking	Time of braking mode, sec	4

Now we will run one numerical experiment and save stress data for the further durability analysis.

1. Load configuration file of the model Vibrostand-configuration.icf (File–>Load configuration)
2. Select the tab FEM | Simulation | Settings, check the field Save modal coordinates and select a location for the results file storage (.tmc and .imc files). By default, the selected direct tasks and flexible subsystem name (Platform.tmc and Platform.imc).

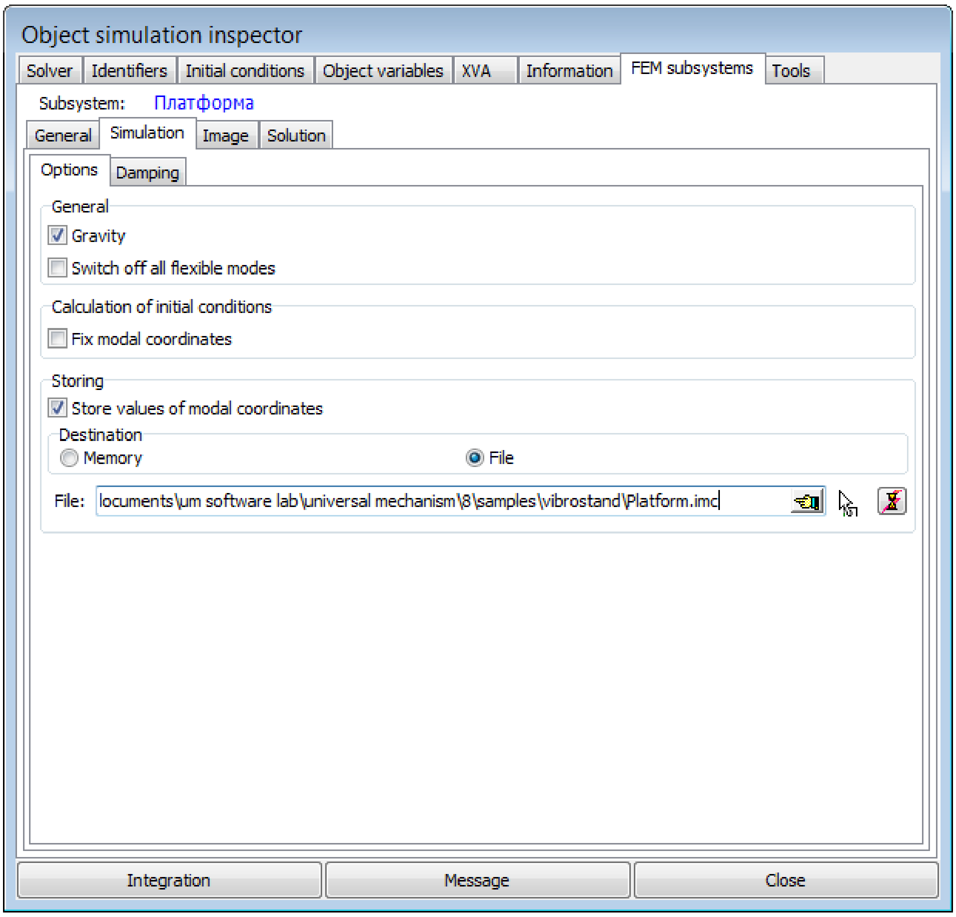


Figure 2.3 Saving history of changes of modal coordinates

3. Select **Solver** tab and start **Integration**[4]. During the numerical simulation *Platform.imc* and *Platform.tmc* files will be written to the directory of the model.
4. Click **Interrupt** in the **Pause inspector** and **Close** in the **Object simulation inspector**.

[4] To make simulation process faster minimize animation window.

2.4. Stress loading analysis

1. From the **Tools** menu select **Durability wizard…**. **Stress loading and durability analysis wizard** main window appears, see Figure 2.4. An empty project will be created automatically.

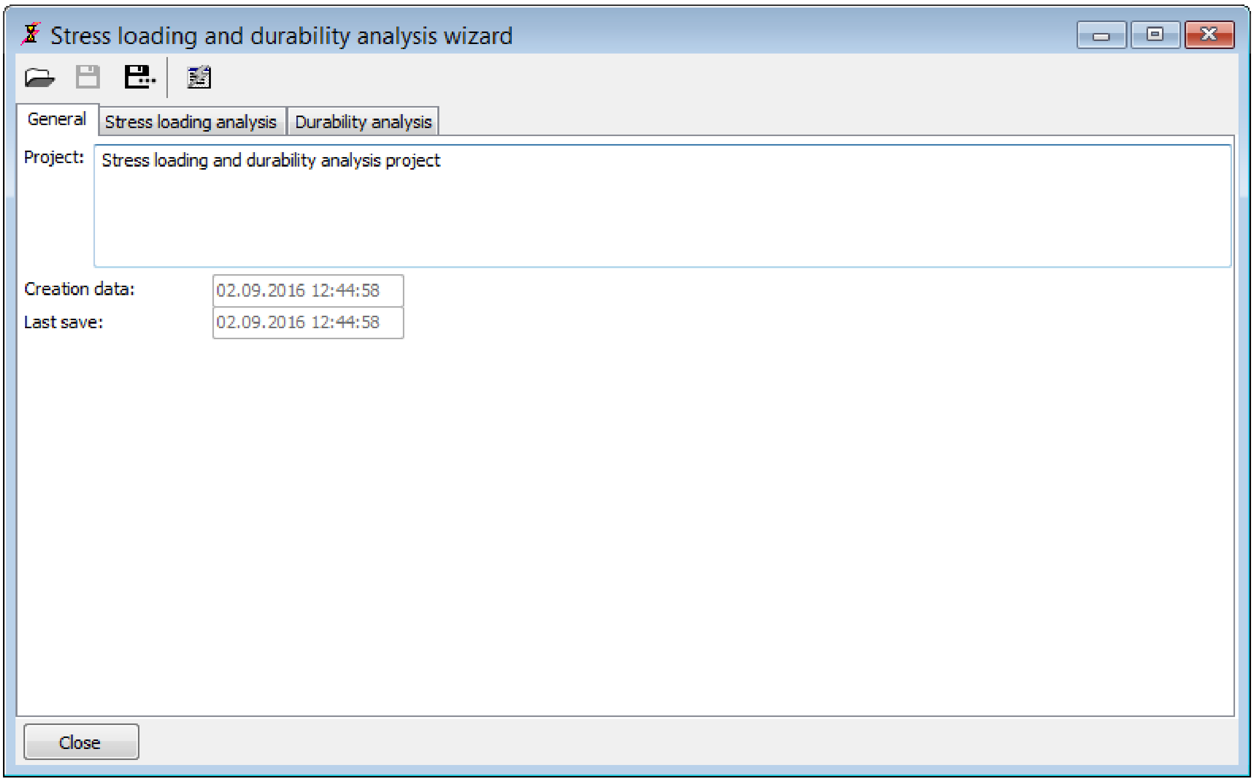


Figure 2.4. Durability wizard window

2.4.1. Load cases description

Operational conditions of the object can be described with the 3 load cases, which can be picked out from the results of the previous dynamics analysis.

1. Select the **Stress loading analysis | Source data | Loading regimes** tab.
2. Click the **+** button to add the *Platform.tmc* file three times, see Figure 2.5. The first loading of the file will take more time than further ones, due to flexible subsystem loading time study.

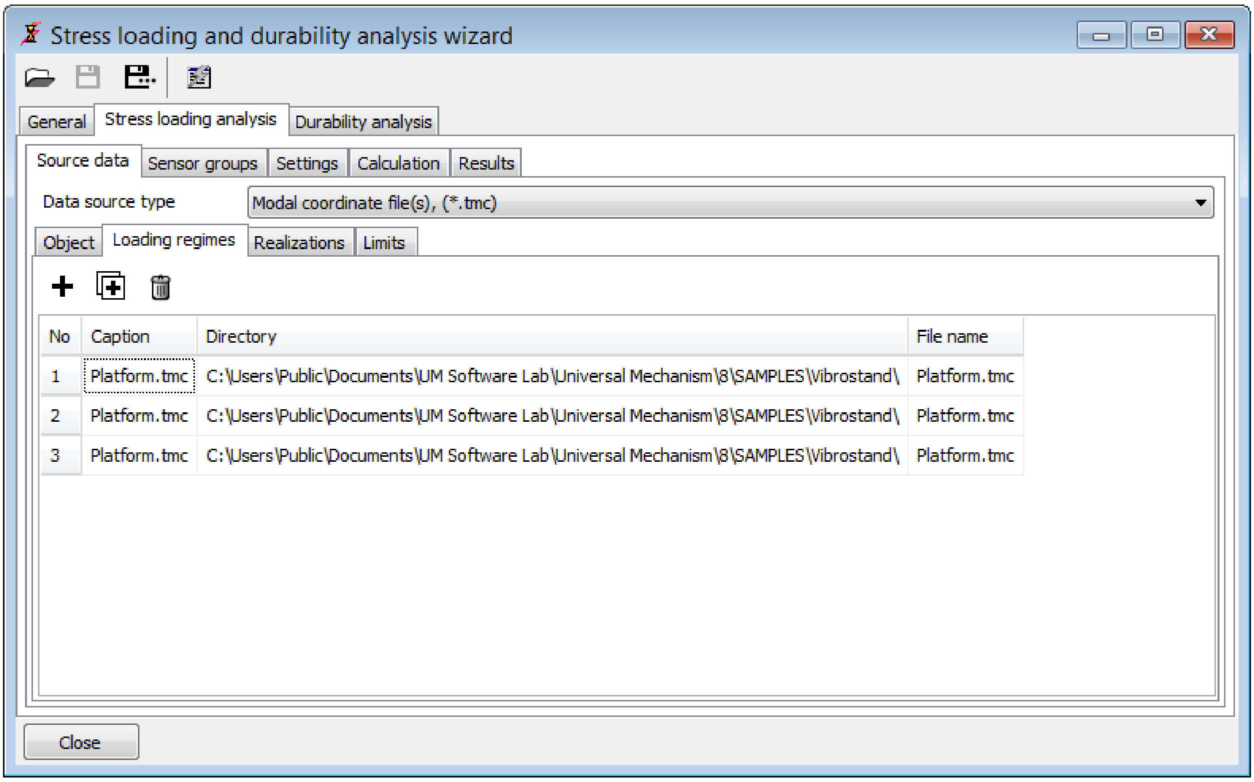


Figure 2.5. Adding of load cases

Load cases captions editing

Let us edit load case captions to make them more definite and clear.

1. Edit captions of load cases in the directly grid as it is shown in Figure 2.6 or load captions from the *LoadCasesCaptions.lcc* file using the context menu, see Figure 2.6.

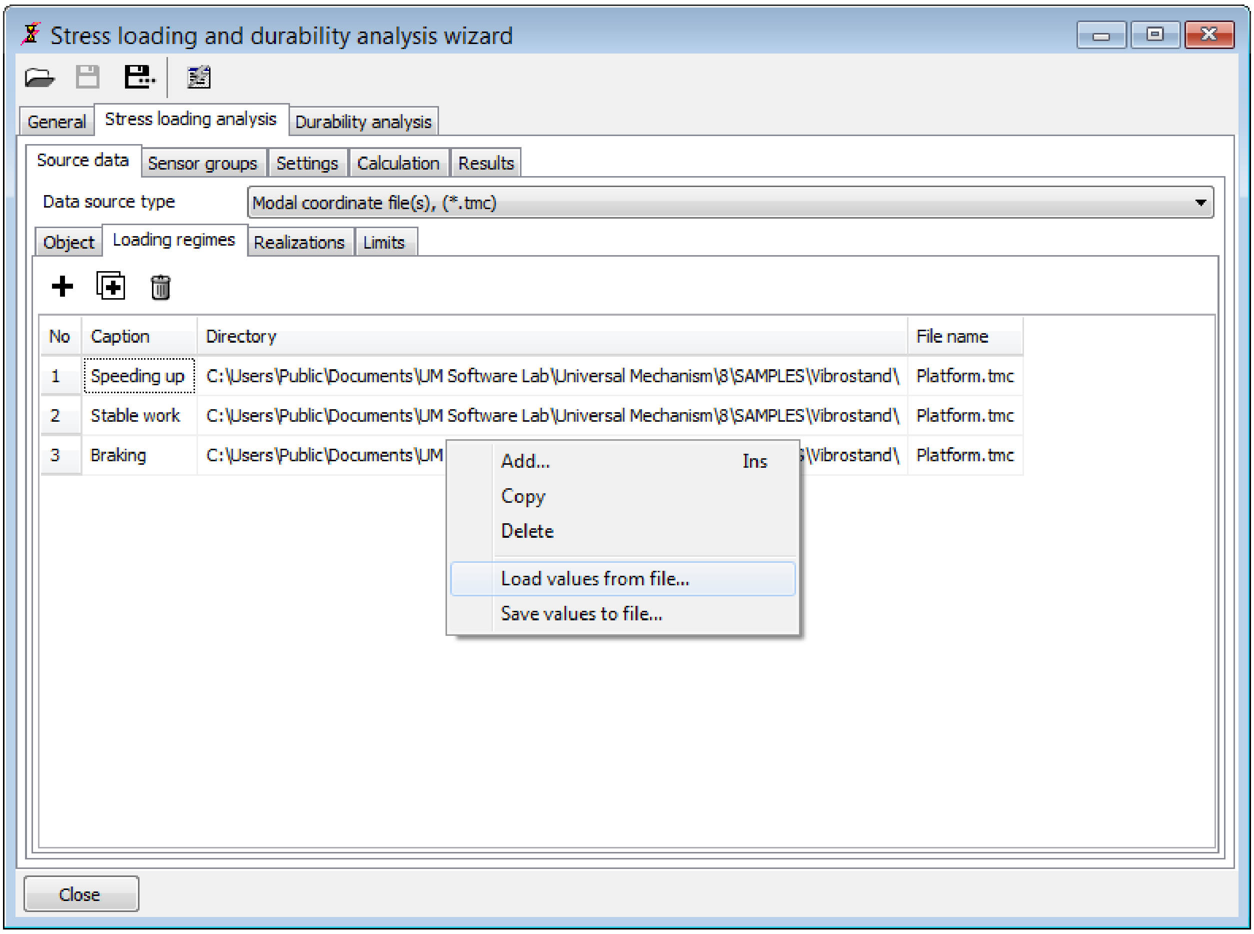


Figure 2.6. Load cases captions editing

Inspecting the object to analyze

1. As soon as you load at least one load case you can see your FEM object at the **Stress loading analysis | Source data | Object** tab, see Figure 2.7.

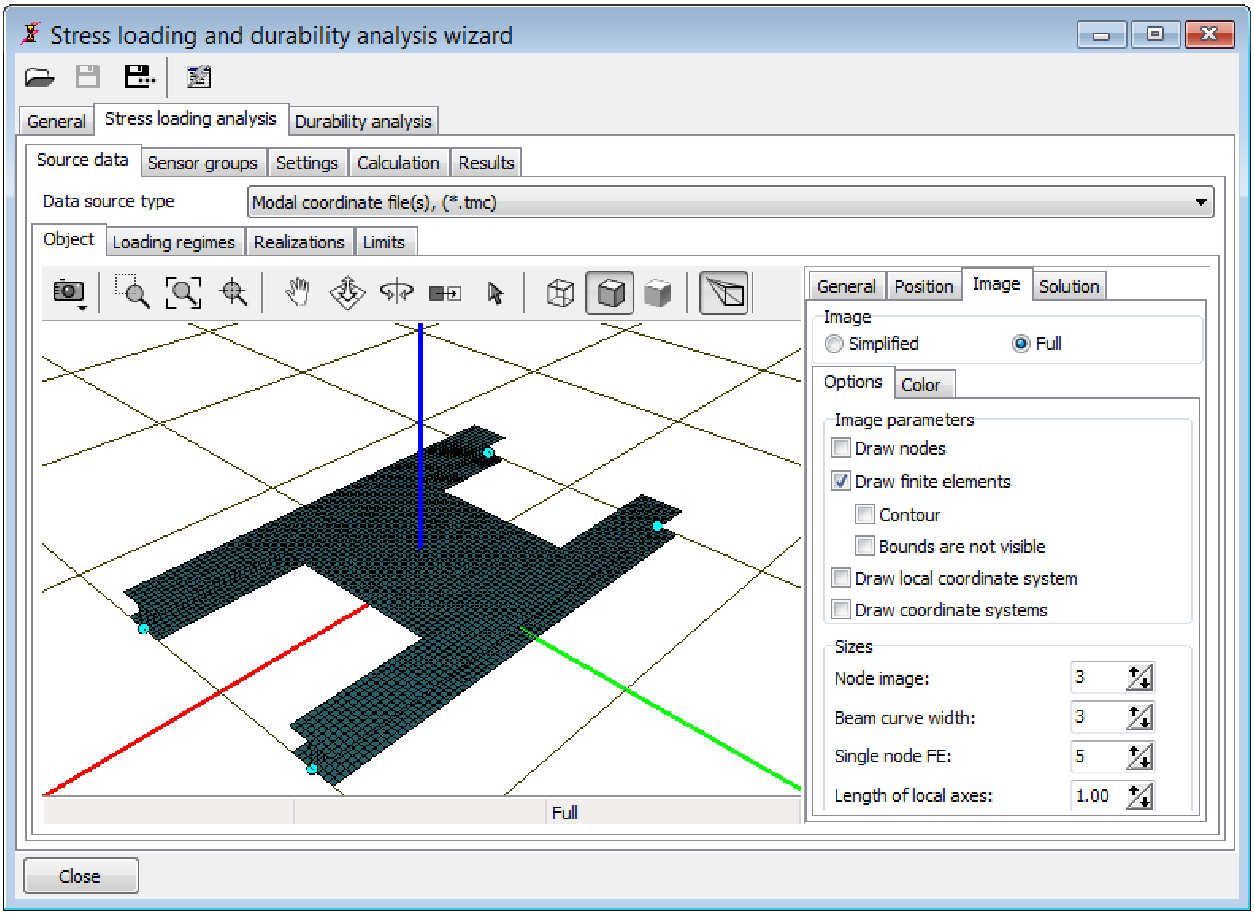


Figure 2.7. FEM model of the frame

Stress time history plotting

Stress histories, used in description of the load cases, must be representative. Let us plot stress time history at the nodes **3773** and **259**[5], for example, see Figure 2.8.

1. Select **Stress loading analysis | Source data | Realizations** tab.
2. Select **Unsigned von Mises by principle stresses** on the left.
3. Set **Node number** to **3773** and **Calculate**. The field  below becomes enable as soon as you calculate stress time history to show.
4. Open a new graphic window and just drag the right bottom field to the graphic window.
5. Repeat 3-4 for **259** node.

[5] Lately we will observe that nodes 3773 and 259 are placed in the most loaded areas of the frame

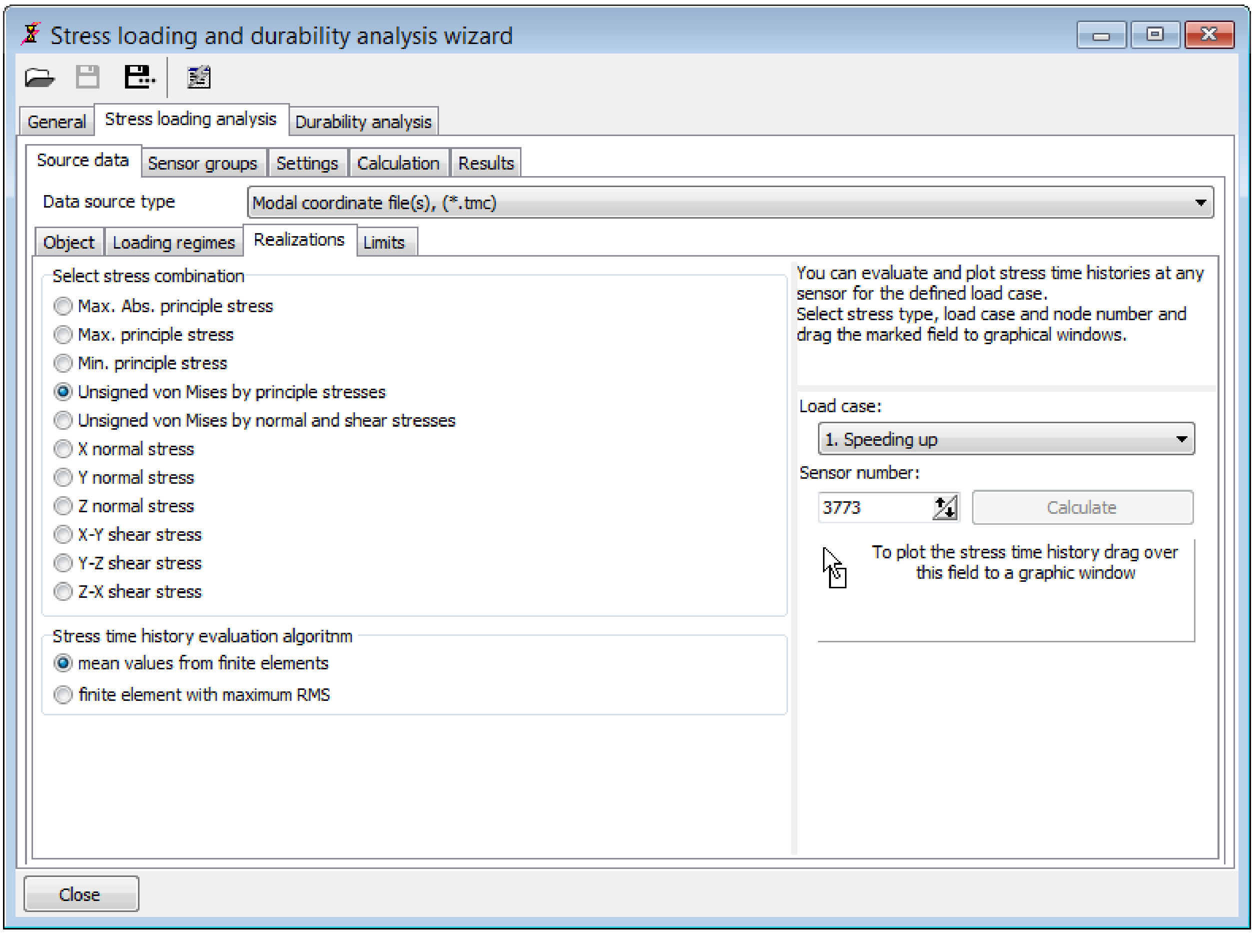


Figure 2.8. Stress time history for node **3773** plotting

You can clearly see well-defined stages of speeding up, stable work and braking modes on the plots, Figure 2.9.

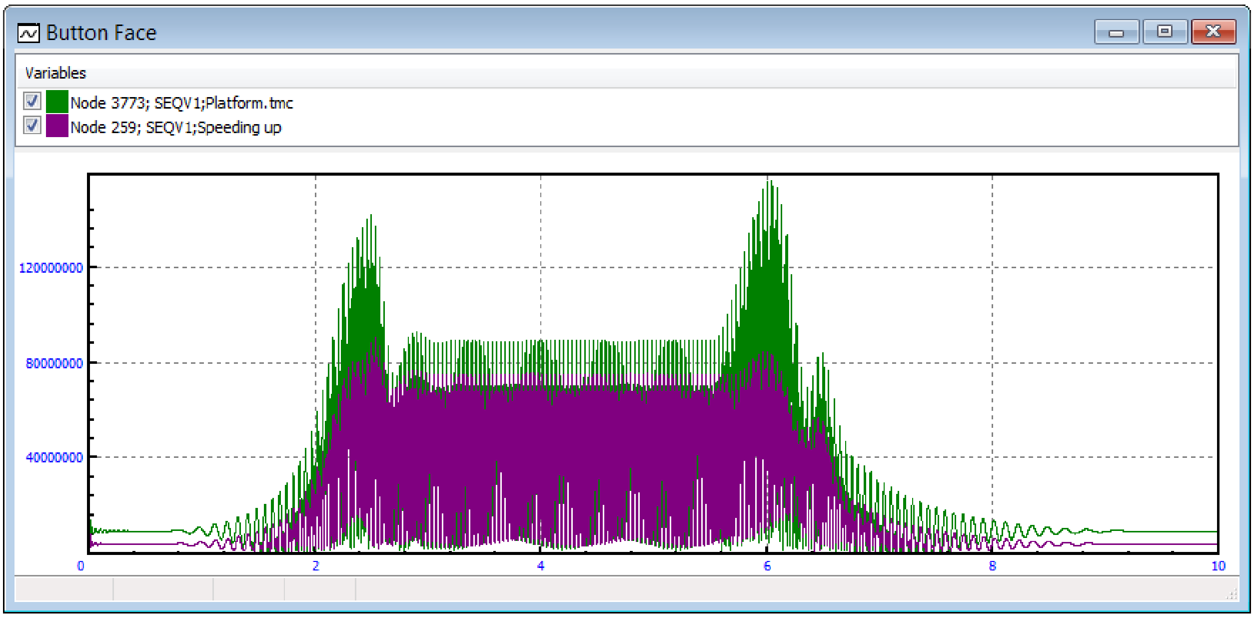


Figure 2.9. Stress time history at the nodes **3773** and **259**

Time intervals of stress histories setting

1. According to Figure 2.9 let us define time intervals for load cases as it is shown in Figure 2.10.
2. You can input values manually or load from the *TimeIntervals.int* file, Figure 2.10.

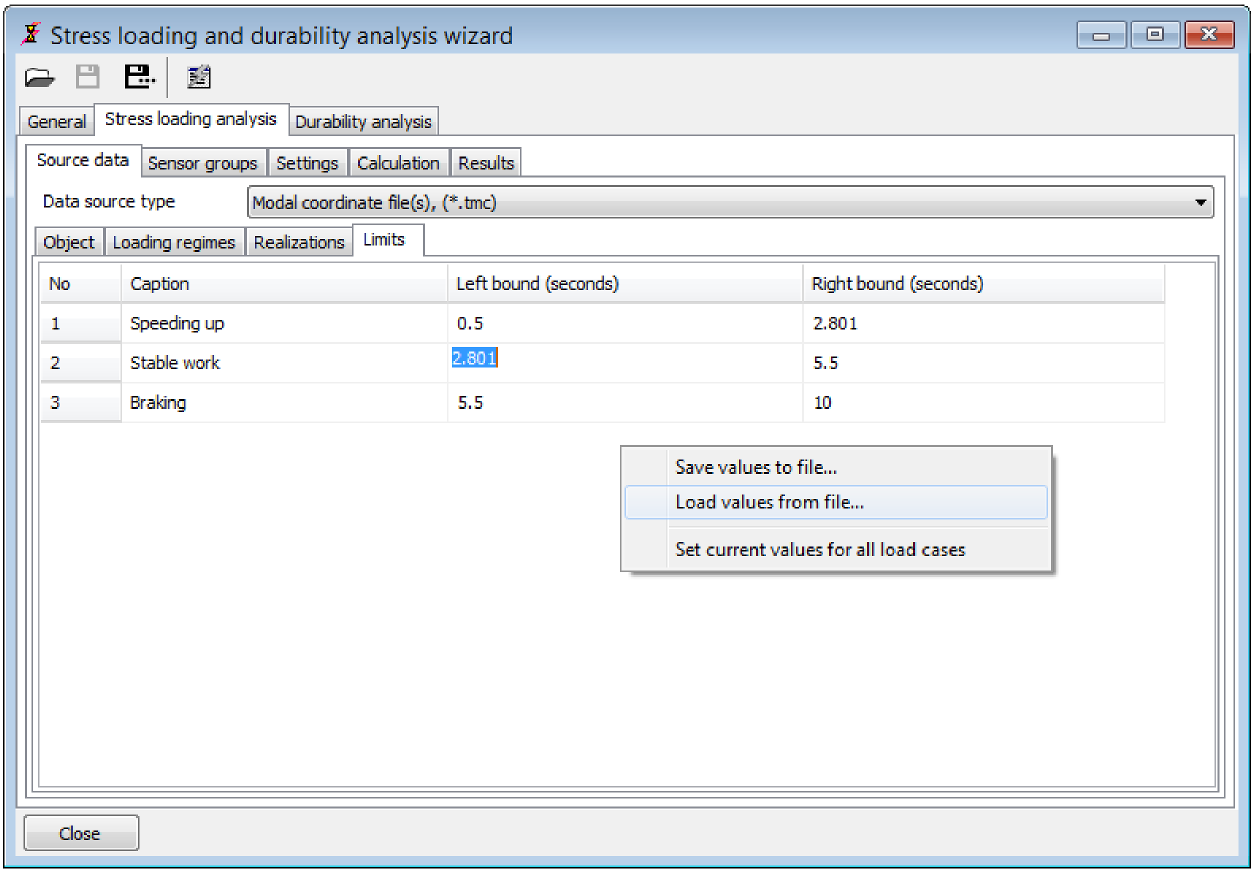


Figure 2.10. Time intervals

2.4.2. Sensor groups initialization

As soon as quantity of finite elements of the frame scheme is not great let fulfill stress loading analysis for all FE-nodes. Therefore, let create sensors at every node of the flexible platform.

1. Select the **Stress loading analysis | Sensor groups**. Node group **All FEM nodes** was created by default, see Figure 2.11.

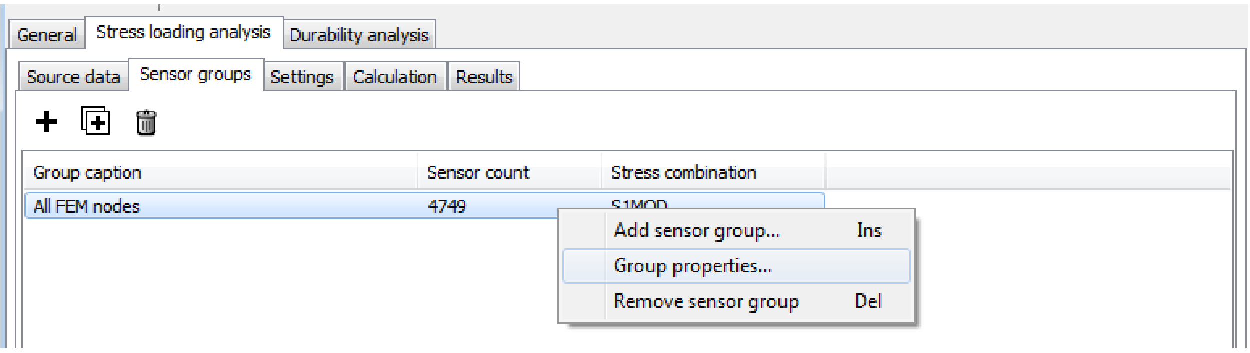


Figure 2.11. Sensor group All FEM nodes initialization

2. Double click on the **All FEM nodes** caption in the list or select the **Group properties…** item of the group popup menu, see Figure 2.11.
3. Sensor group properties window will be opened. Select the **Node list** tab to look at node list of the group, see Figure 2.12.

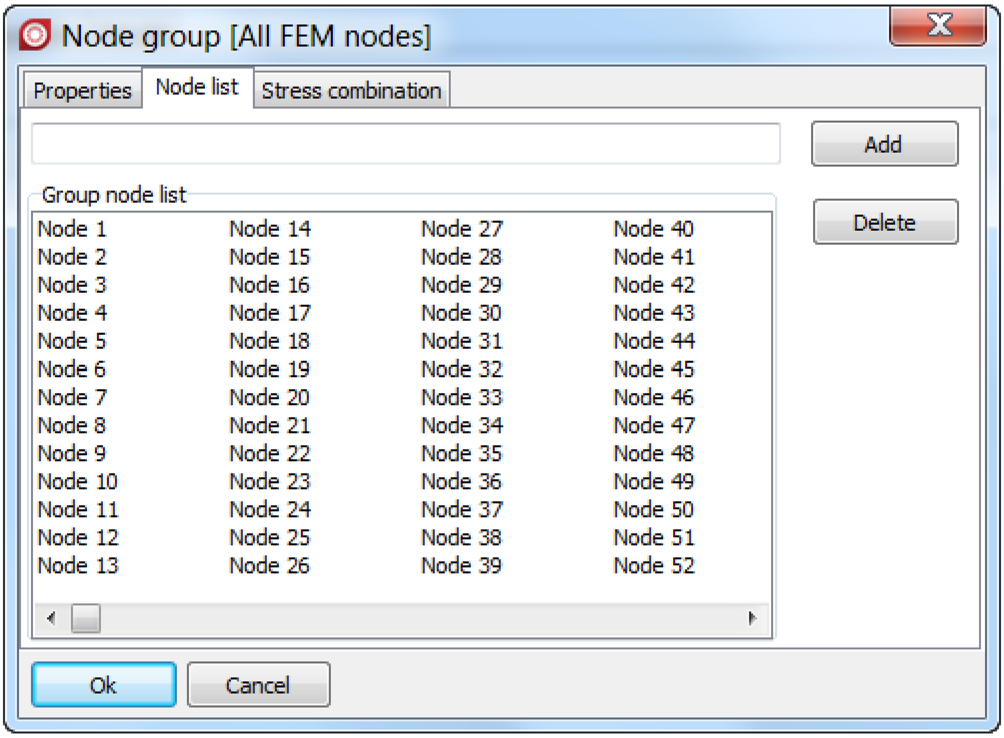


Figure 2.12. Node/sensor list of the **All FEM** data group

Stress combination

4. Select the **Stress combination** tab, see Figure 2.13.
5. Set the **Select stress combination** field to the **Unsigned von Mises by principle stresses**.
6. Set **Stress time history evaluation algorithm**[6] to the **mean value from finite elements**.

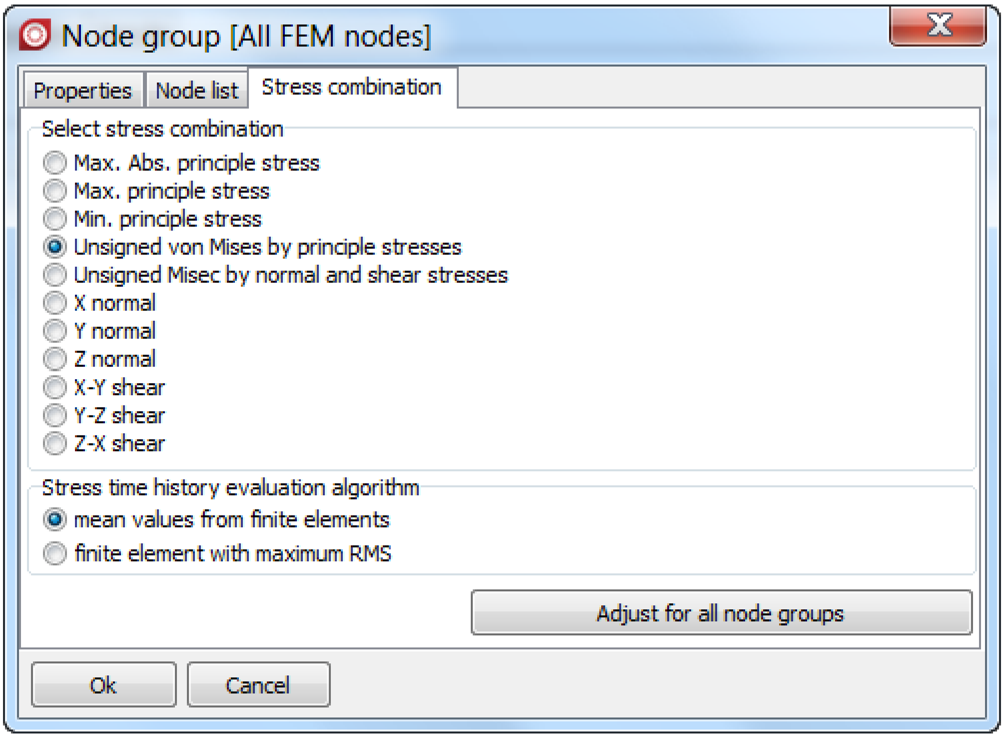


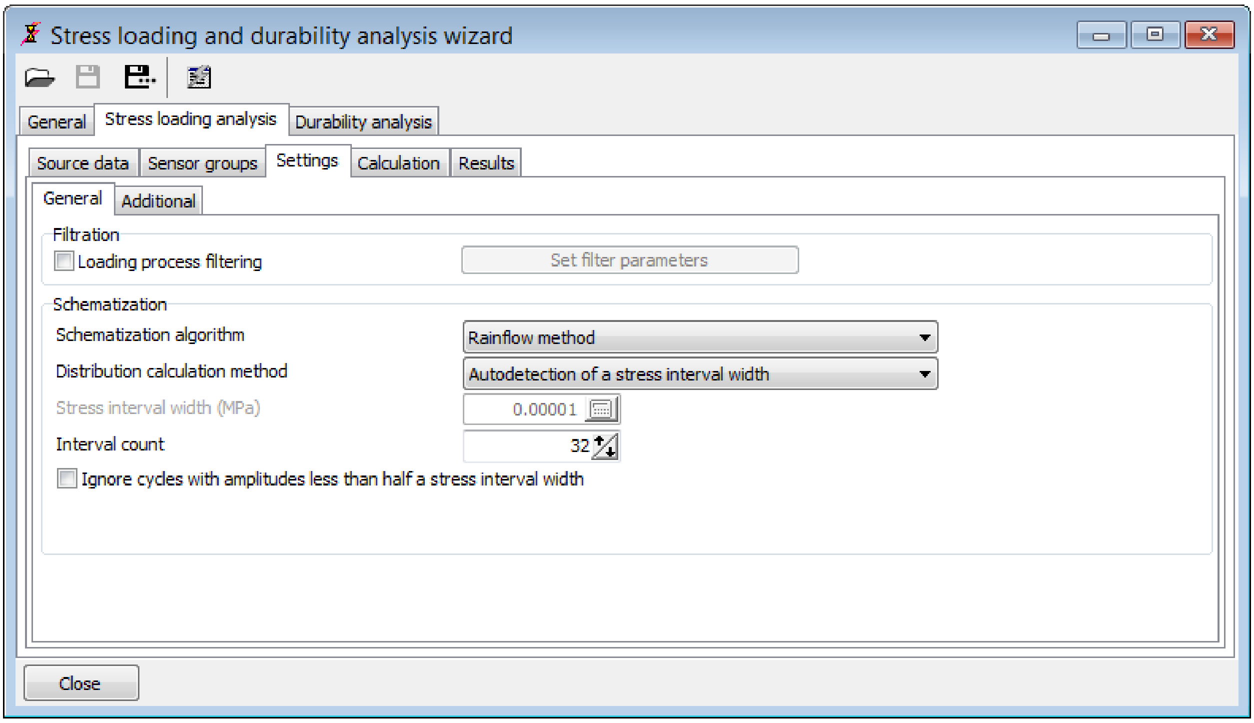
Figure 2.13. Stress combination setting for **All FEM nodes** group of sensors

7. Click **Ok** to save changes to **Node group [All FEM nodes]** description.

[6] Every node of FE mesh generally can belong to several finite elements. This option indicates how to calculate stresses in nodes when they belong to several finite elements

2.4.3. Stress load evaluation settings

1. Select tabs **Stress loading analysis | Settings | General** and **Stress loading analysis | Settings | Settings for carriage-building method**, and set the following evaluation options, see Figure 2.14.



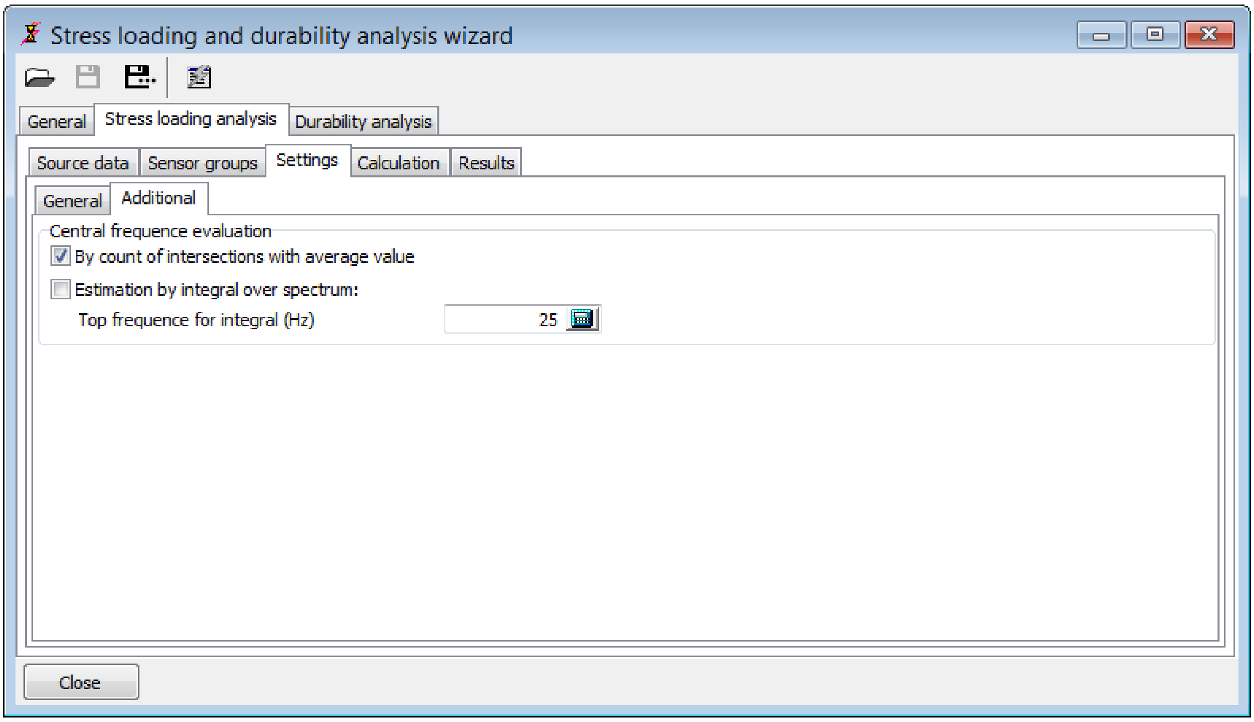


Figure 2.14. Stress loading evaluation parameters

2.4.4. Save project to file

1. Use the  bottom in the top left corner of **the Stress loading and durability analysis wizard** to save project to file.

2.4.5. Stress loading data calculation

1. Select **Stress loading analysis | Calculation** tab.
2. Click the **Calculate** button.

Source data verification will be fulfilled automatically. Then the stress loading evaluation will start.

Progress bar indicate stages of the calculation process, see Figure 2.15. It might takes about 10-30 minutes depending on computational power of your computer[7].

Use the bottom placed more to the right from main status bar to display bars of additional data.

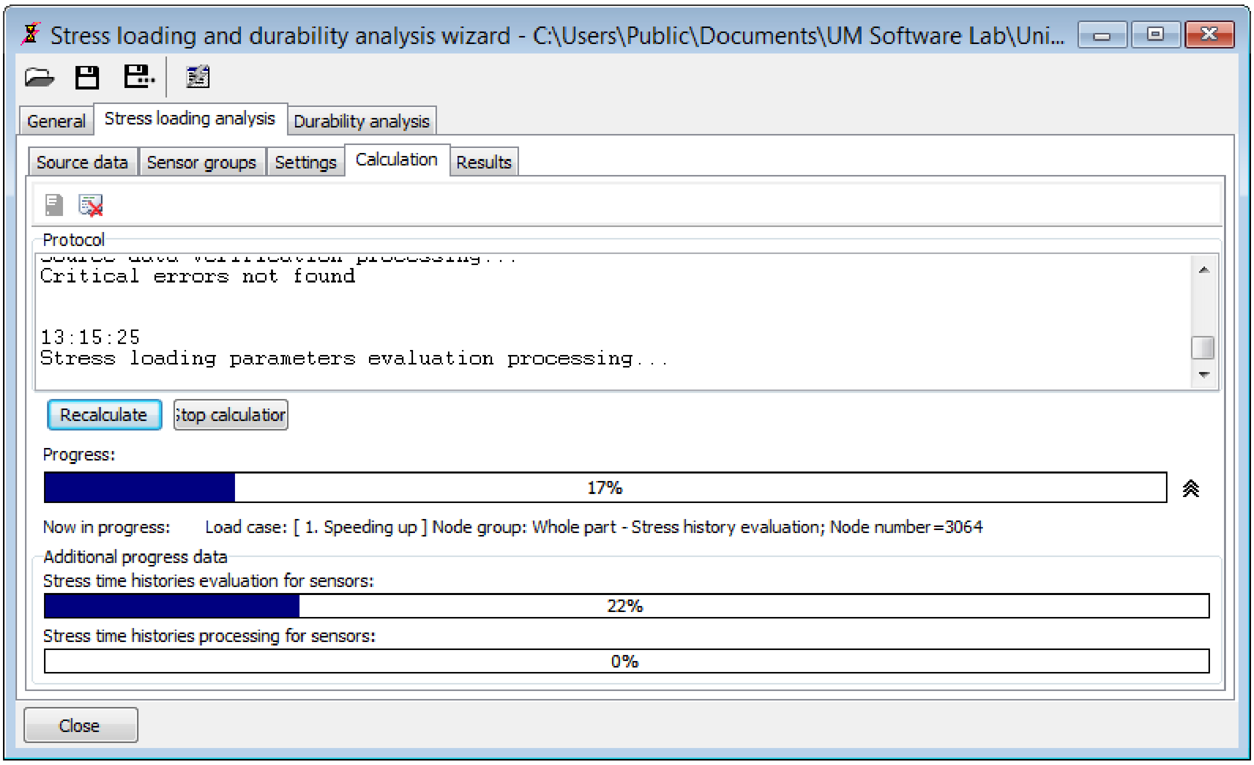


Figure 2.15. Stress loading calculation

[7] It is high time to have a cup of coffee, is not it?

2.4.6. Stress loading calculation results analysis

1. Select **Stress loading analysis | Results | Sensor list** tab.
2. In the **Load** case select **Combined stressload block**.
3. To sort nodes by decreasing of maximal amplitude values click on the header of the correspondent column, as it shown in Figure 2.16.

You can select list columns to display with the help of the popup menu. All columns are displayed by default.

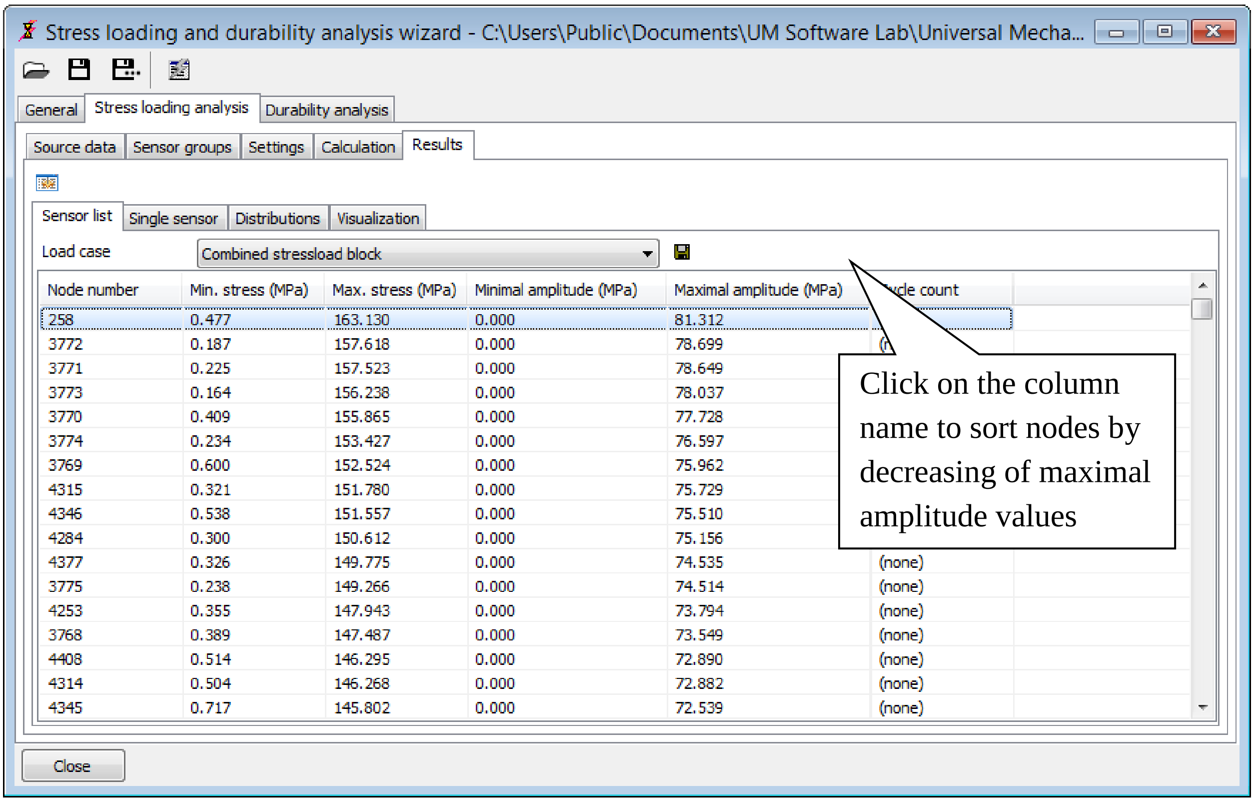


Figure 2.16. Results: node list

You can see that maximal stress amplitude values resulted in fatigue damage of material are indicated at the nodes 258, 3772, 3771, 3773.

4. Repeat steps 1-3 to display results for the particular load cases. Make sure that maximal stress amplitude values are indicated in the following nodes:

Speeding up mode – nodes **258, 3770, 3771, 3769.**
Stable work mode – nodes **258, 542, 3769, 3770.**
Braking mode – nodes **258, 3772, 3771, 3773.**

Let us show the position of these nodes and total stress loading parameters distribution at all FEM nodes.

5. Select **Stress loading analysis | Results | Visualization** tab.
6. Set **Load case** to **Combined stressload block**.
7. Set **Select data for visualization** to **Maximal values of stress cycle amplitudes (MPa)**, see Figure 2.17.
8. Click **Show**. New animation window with distribution of maximal values of stresses appears, see Figure 2.18.

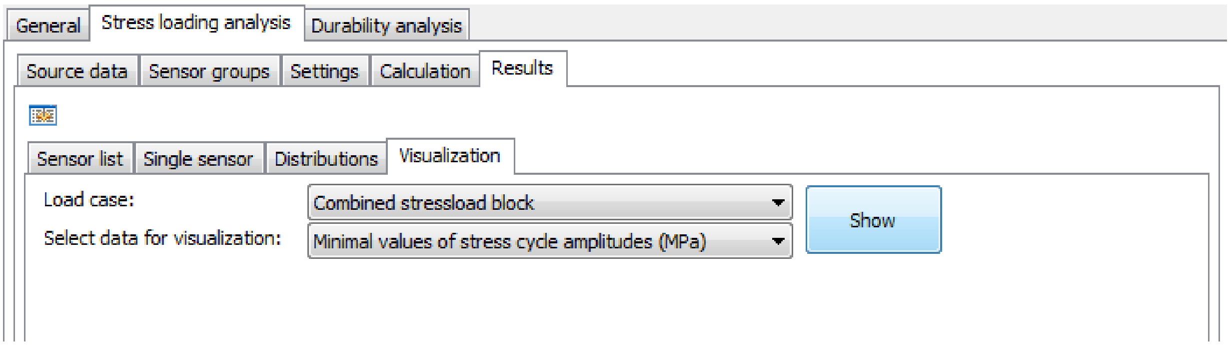


Figure 2.17. Results: visualization

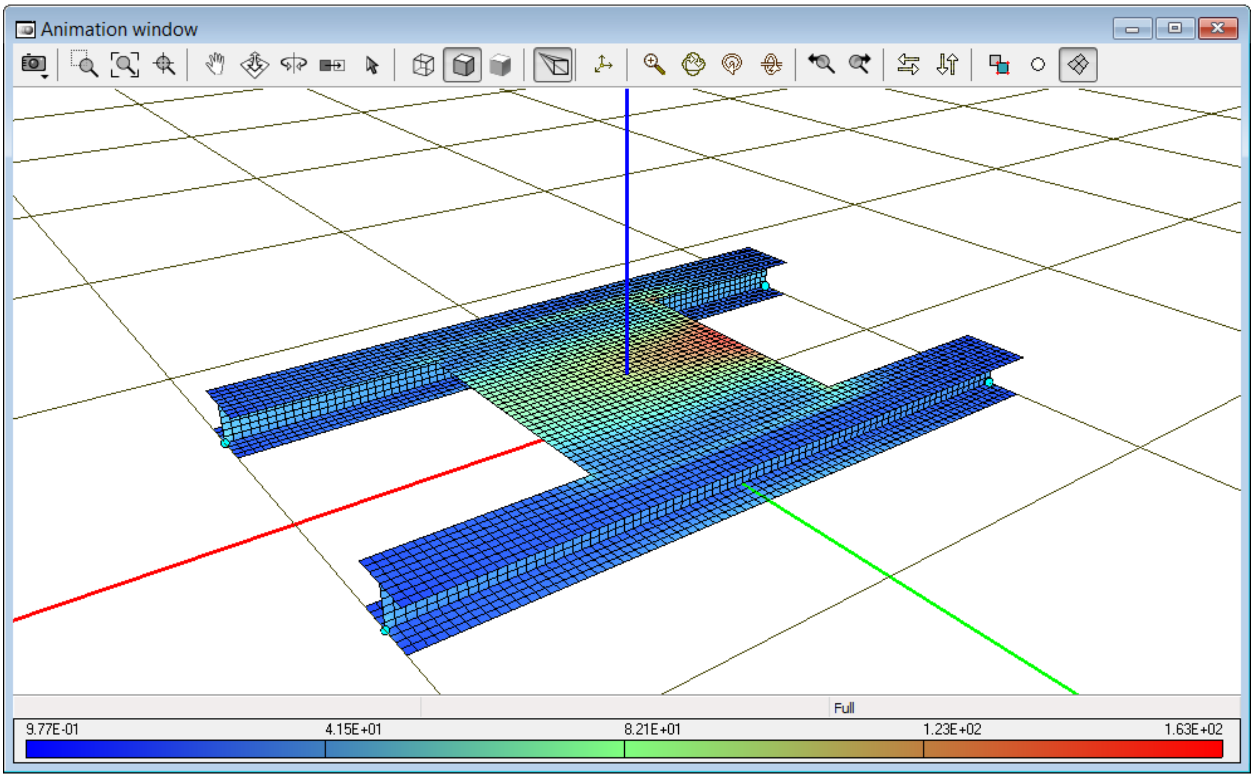


Figure 2.18. Distribution of maximal amplitudes

9. Click on the button at the top panel of the animation window to display nodes.
10. From the context menu of the animation window select the **Select FEM nodes** menu command. New dialog window appears.

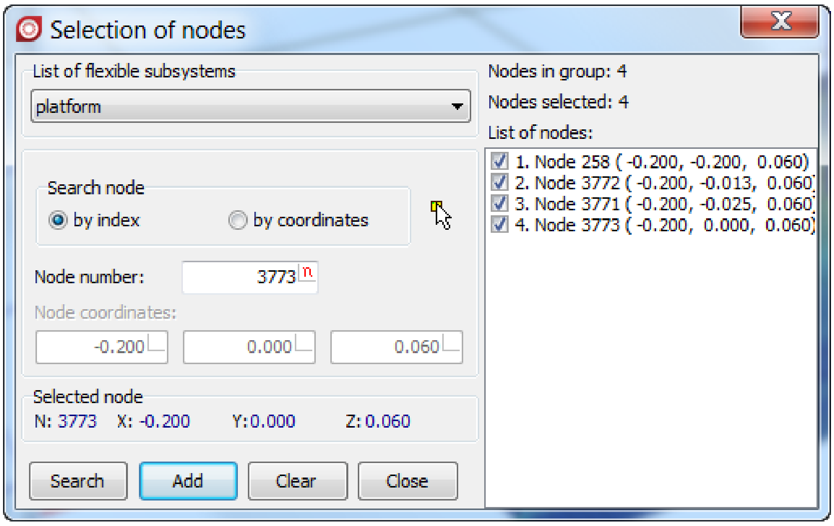


Figure 2.19. Select node to highlight

11. Select the **by index** option and set **N** to **258**, see Figure 2.19.
12. Click **Search** button to find the node and then **Add** button to add it to the list of highlighted nodes.
13. Repeat the 10-11 steps for nodes **3772, 3771, 3773**.

14. Click **Close**. Selected nodes will be highlighted at the animation window with green, see Figure 2.20. Guide cursor to point to display node number and its position.

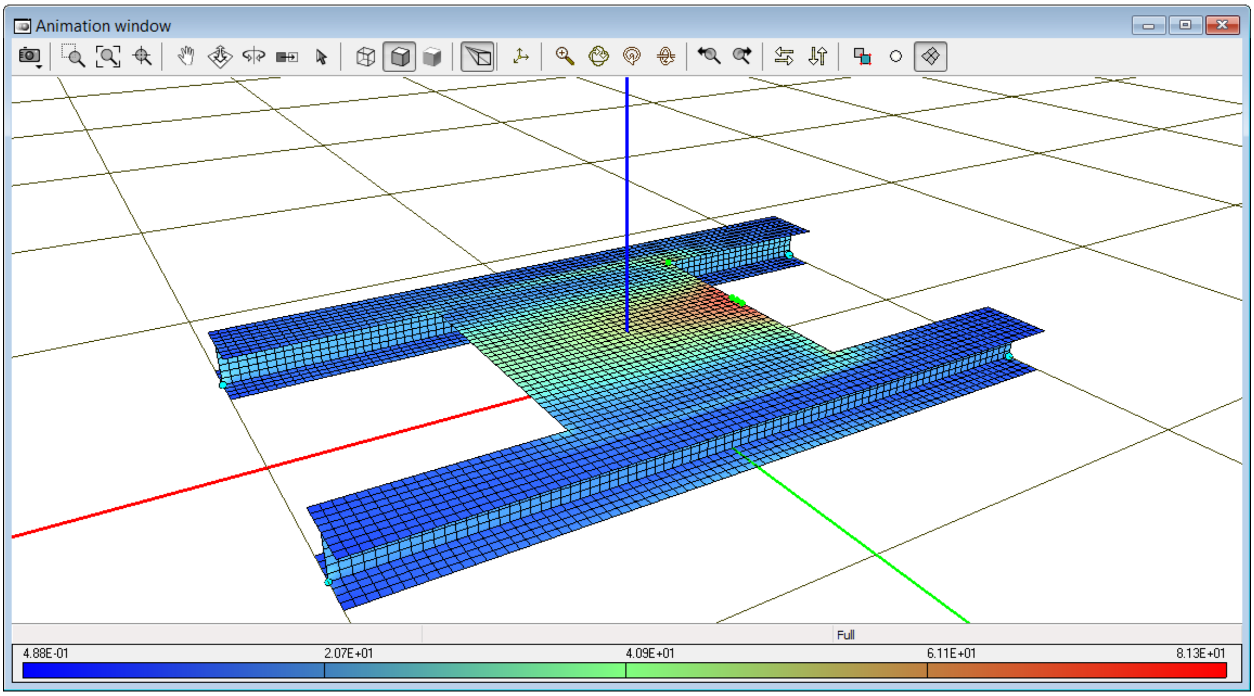


Figure 2.20. Highlighted nodes in the animation window

Let us see, which of the load cases results in maximal amplitudes of stress cycles at node 3117.

15. Select **Stress loading analysis | Results | Single sensor** tab and set **Node** number to **258**, see Figure 2.21.

Speeding up and **Braking** load cases result to the highest maximal amplitude values at the node.

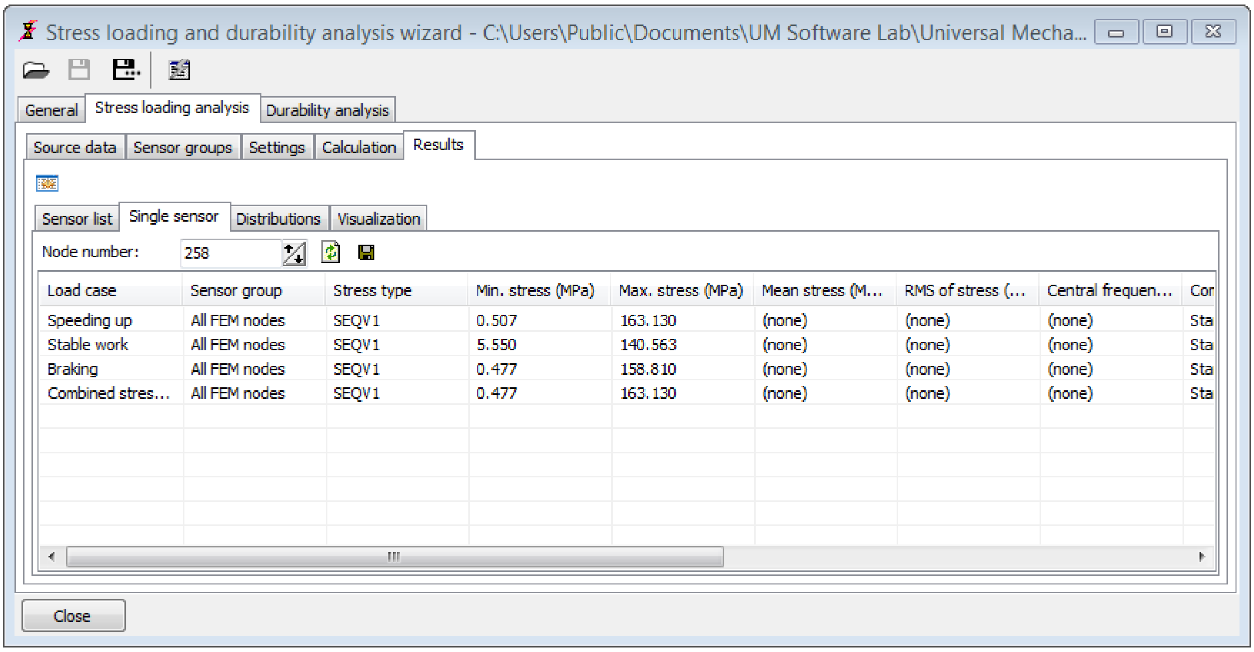


Figure 2.21. Results: particular node

16. To inspect stress loading parameter distributions for the load cases at the node select **Stress loading analysis | Results | Distributions** tab. Input node number and select the load cases to display single and two-parameters distributions of amplitude and mean values in table form, see Figure 2.22.

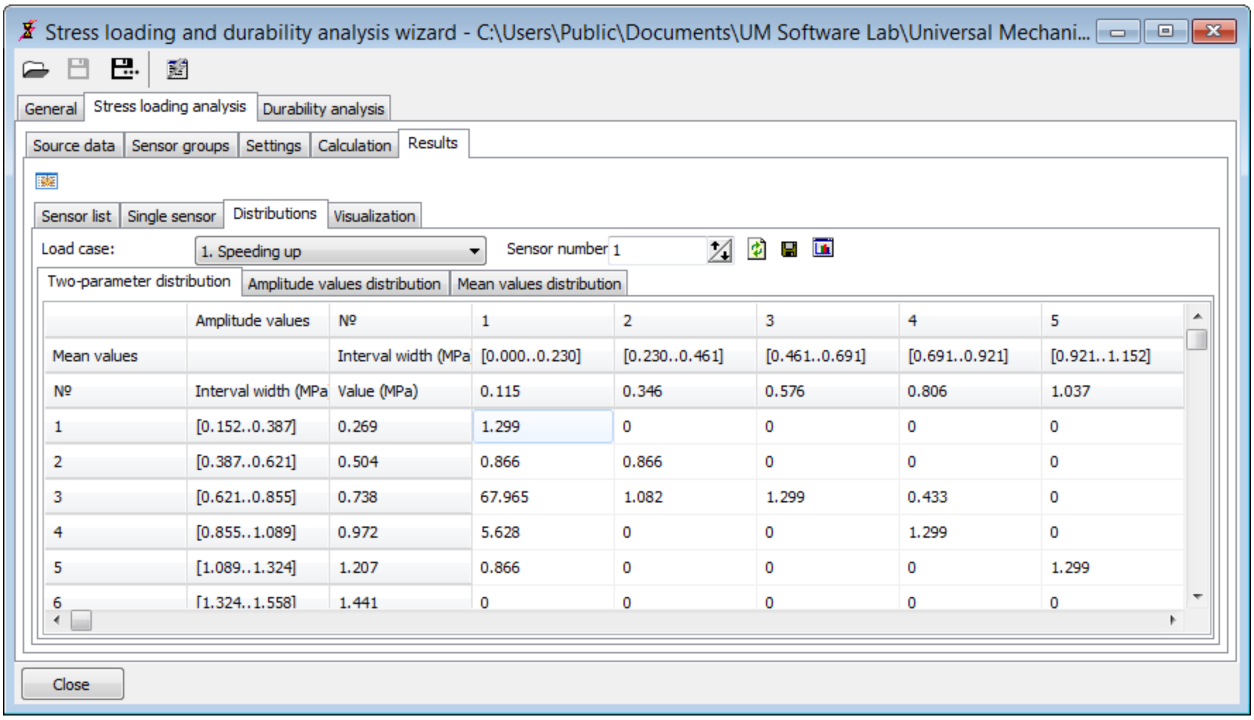


Figure 2.22. Results: stress loading parameters distributions (table form)

17. To plot the distributions click the button. Specialized window includes plots of data, shown at the tables, see Figure 2.23.

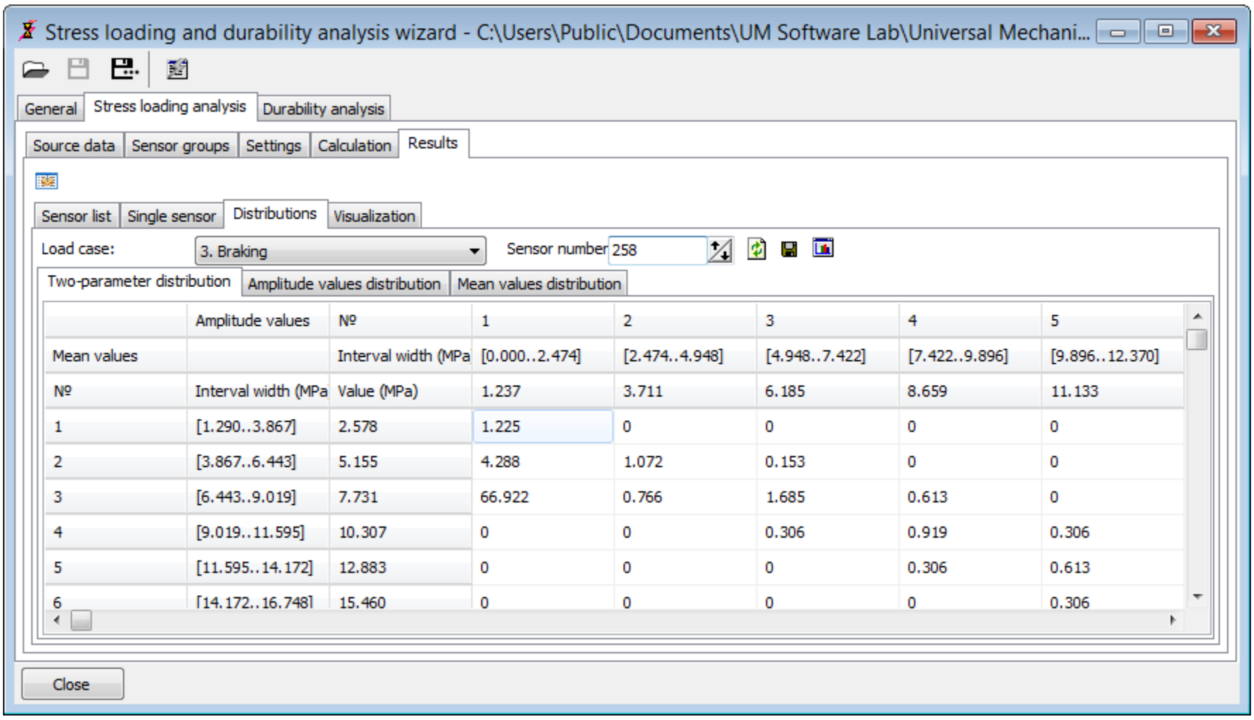


Figure 2.23. Results: stress loading parameters distributions (diagram)

The results of stress loading analysis can be used as source data for durability analysis. It can be fulfilled in some external programs, or one can use functionality of **Durability analysis** instruments realized in the **Wizard**. These instruments give a possibility of fatigue strength and durability estimation in accordance to the common machine-building and road vehicles methodic.

2.4.7. Save project to file

1. Use the bottom in the top left corner of the **Stress loading and durability analysis wizard** to save project to file.

2.5. Durability analysis

2.5.1. Durability analysis method initialization

1. Select **Durability analysis | Evaluation method** tab and set **Evaluation method** to **S-N method**, see Figure 2.24. This method is the general method of S-N analysis realized in **UM Durability**.

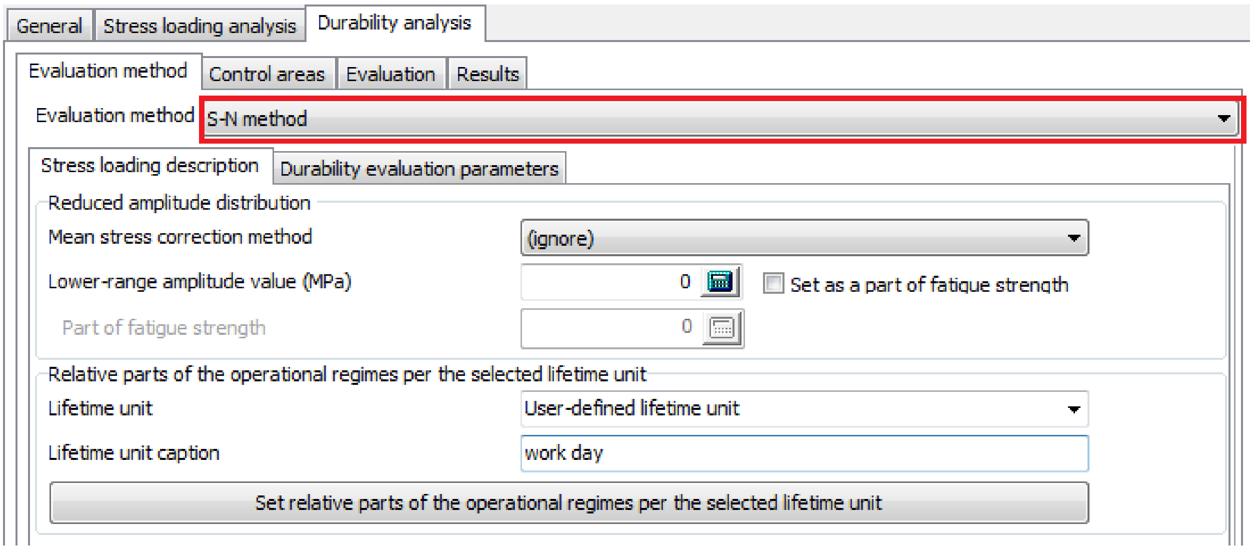


Figure 2.24. Parameters setting: stress loading data description

2. Select the **Stress loading description** tab, see Figure 2.24.

For this first introduction to **UM Durability** let us ignore influence mean stress values to fatigue damage.

3. Set **Mean stress correction method** to (**ignore**).

Zero **Lower-range amplitude value** means that all the loading cycles, independently on their parameters values, will be taken into account in the procedure of durability parameters evaluation.

For the durability evaluation we will use **the User defined life time unit – workday**.

4. Set **Life-time unit** to **User defined**.
5. Input **workday** in the **Life-time unit caption** box, see Figure 2.24.
6. Click «**Set relative parts…**» button to define number of repetitions of the previous defined load cases per a workday.

Let the motor operates continuously in working mode 8 hours per a day, 310 days per a year. Every day it is started one time and stopped one time.

Accordingly, repetition numbers for **Speeding up** and **Braking** load cases per a workday equal **one. Stable work** load case represented with the 2.7 second stress time history (duration is evaluated from the time interval of the load case, see Sect. 1, page 10). Eight-hour workday means that repetition number for **Stable work** load case equals (8*60*60 seconds per day) / 2 = **10670**. Enter repetition number values to cells of the table as it shown in Figure 2.25.

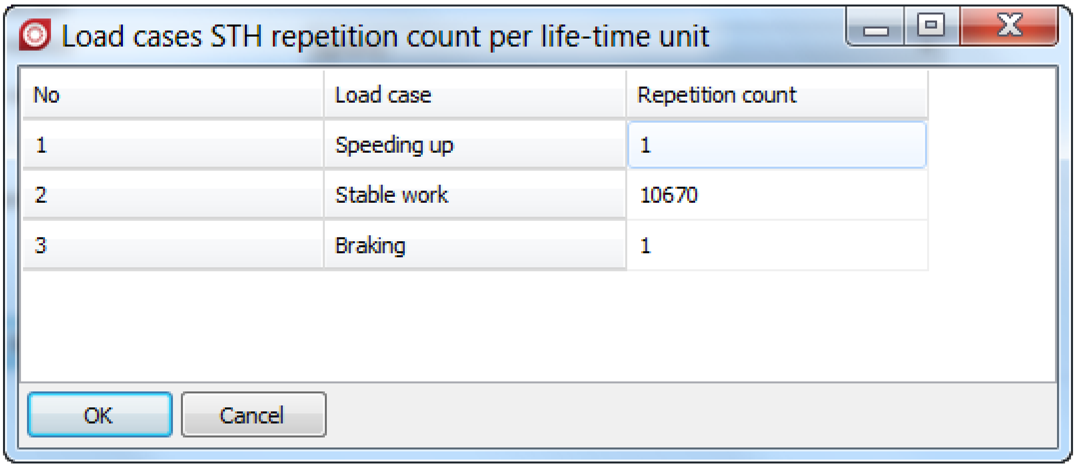


Figure 2.25. Repetition numbers of the load cases setting

7. Select **Durability evaluation parameters** tab, see Figure 2.26.

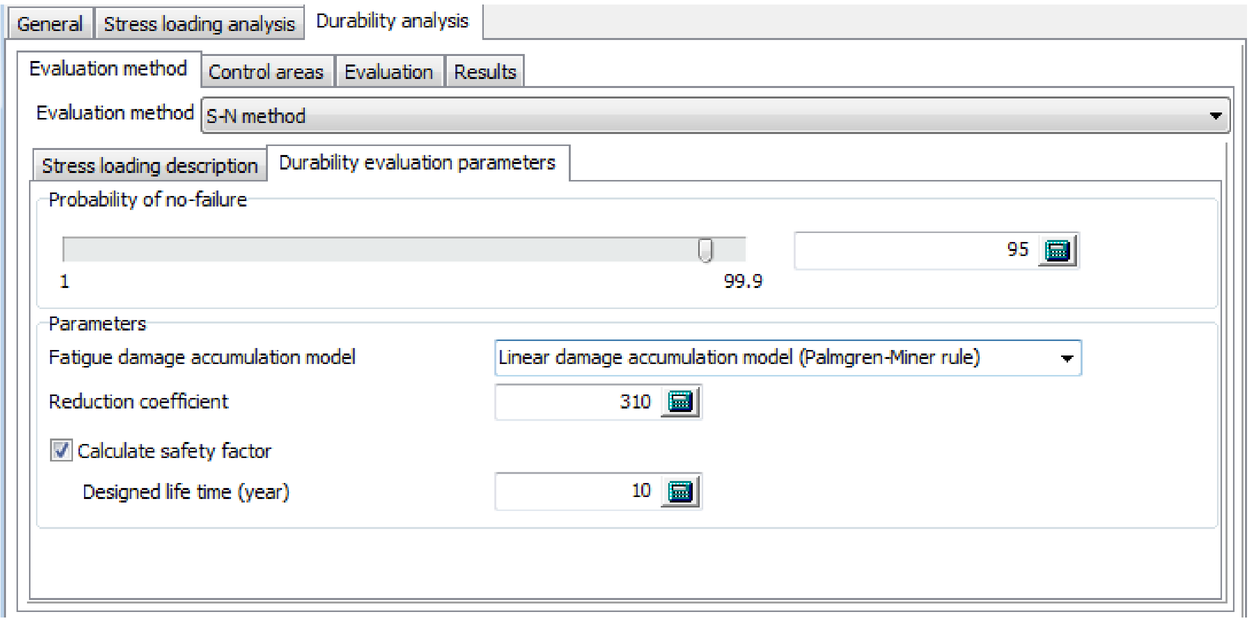


Figure 2.26. Parameters setting: durability evaluation parameters

8. Set **Probability of no-failure** to **95%**.
9. Set **Reduction coefficient** to **310** (310 workdays a year).
10. Turn on **Calculate safety factor** control and set **10** years for **Design life time value**.

2.5.2. Control area selection

Selection of control areas for the durability analysis is based on these results, see Sect. 2.4.6, and construction features of the platform.

Maximal stress cycle amplitude values are observed in a middle part of the top plate (nodes **3773**) for the **Speeding up** and the **Braking** load cases; and at the connections of the top plate and the supporting beams (nodes **258, 542**). Corners of the connections are dangerous according to the influence of stress concentration factor to fatigue resistance of the object.

FEM of the gives fine estimation of stress shape of the middle part of the top plate, but results for connection areas can have some inaccuracy. Therefore estimation of fatigue strength of the top plate will be fulfilled for local stresses, and fatigue strength of the connection areas will be hold by nominal stresses.

Description of control areas and fatigue resistance properties

Let us set initialize the control areas.

1. Select **Durability analysis | Control areas** tab, see Figure 2.27.

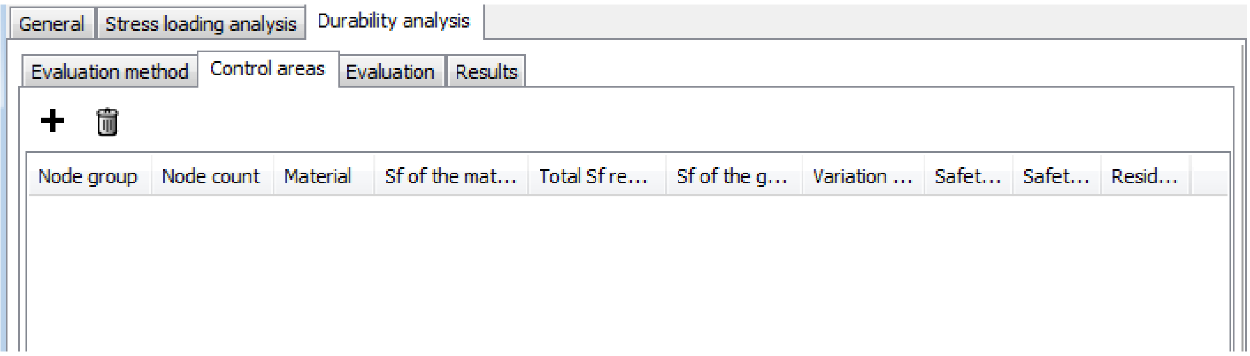


Figure 2.27. Control area list

2. Add a new control area. The control area properties description form will be appeared, see Figure 2.28.
3. Change control area caption to the «**Top plate**».

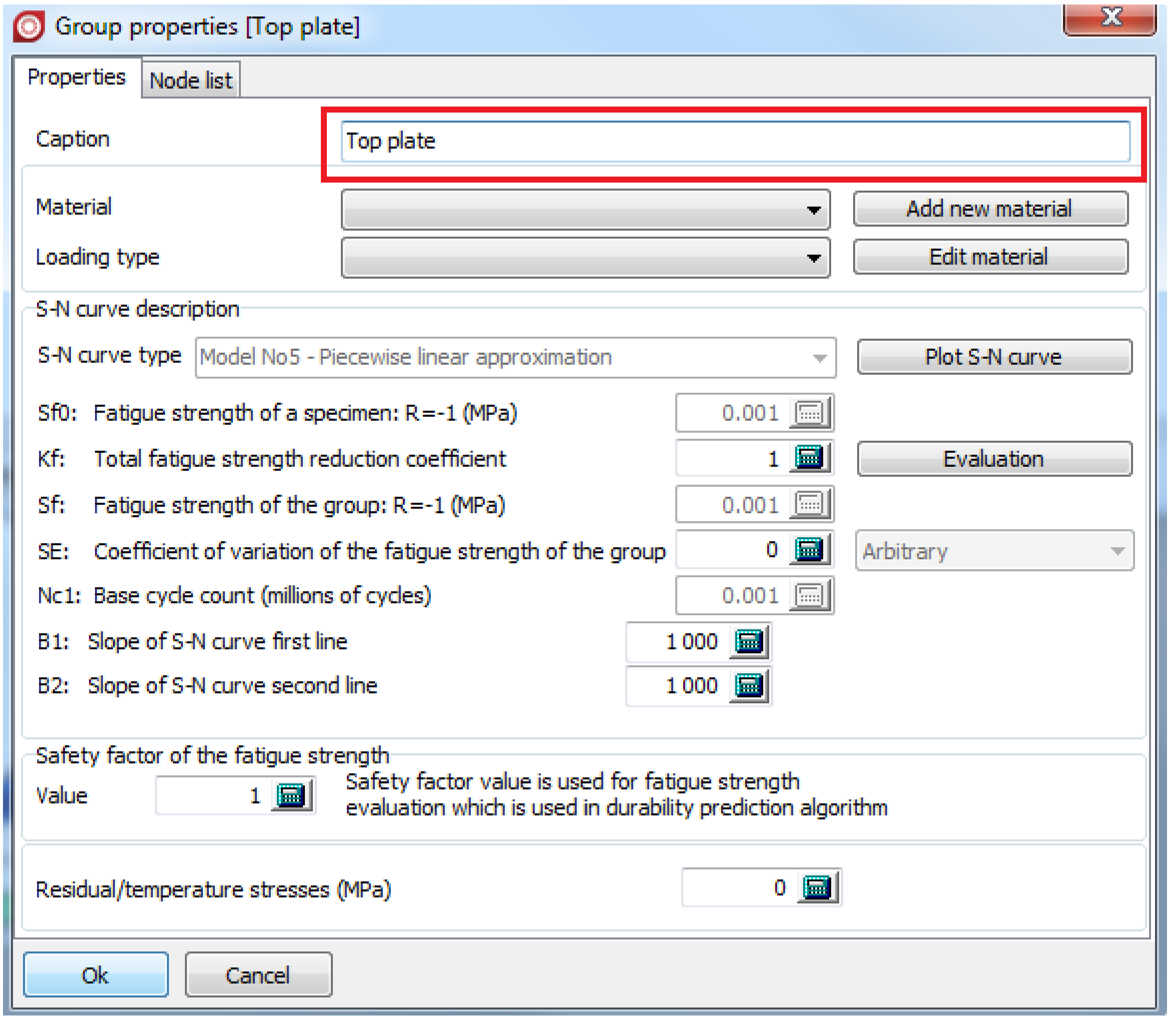


Figure 2.28. Control area properties window

4. Select **Node list** tab.
5. Load node list from the file *Top plate node list.nls* (use the popup menu command **Load node list from file**), see Figure 2.29.

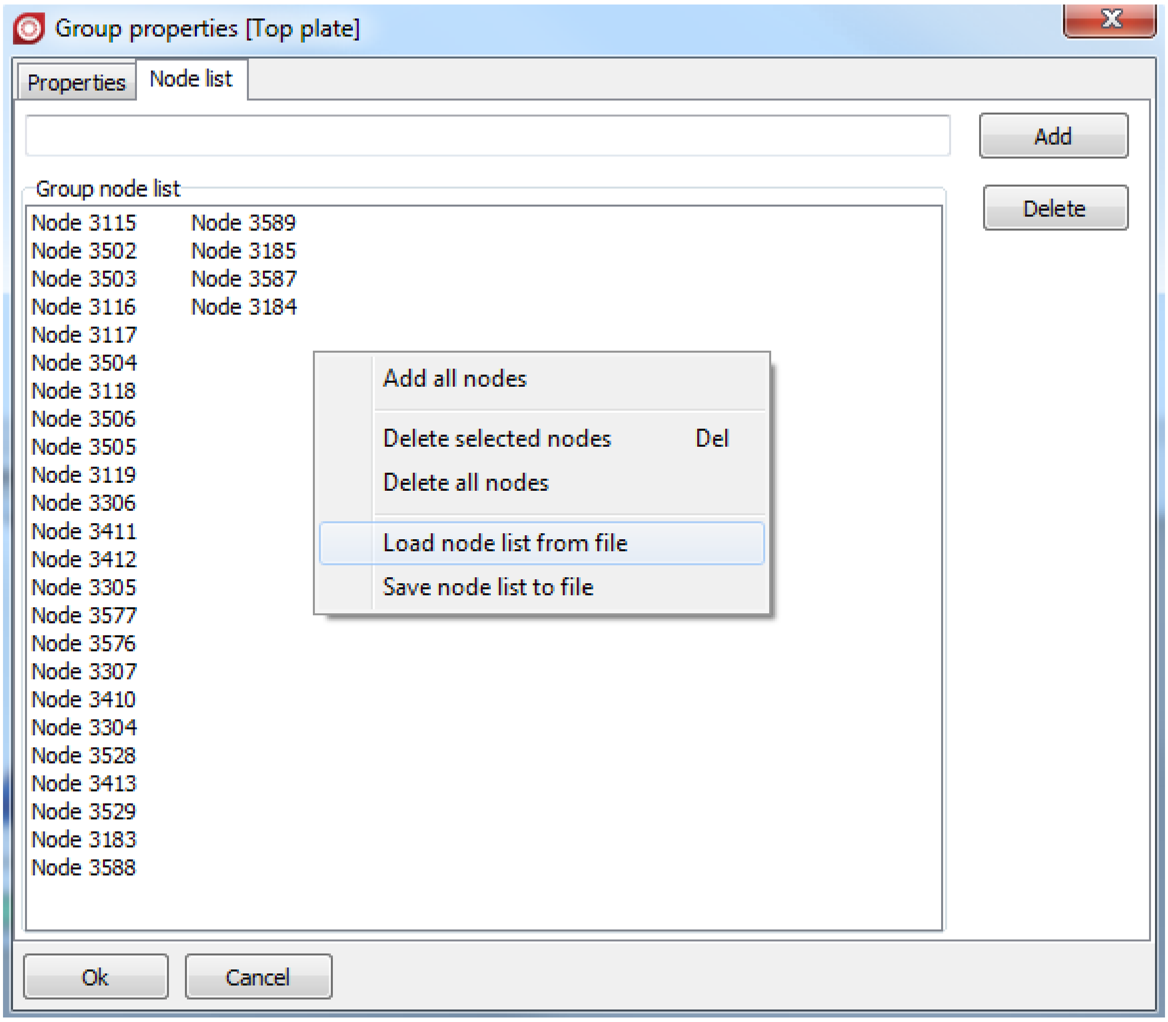


Figure 2.29. Node list initialization for the **Top plate** control area

To describe fatigue resistance of the group we should define material of the platform.

6. Come back to the **Properties** tab and click the **Add new material** button (see Figure 2.28) to define the following material properties:

Static strength properties		
Sy:	Yield strength (MPa)	305
Su:	Ultimate strength (MPa)	440
Fatigue resistance properties for a standard specimen of the material		
Sf:	Fatigue strength, which accords to 50 percentage fracture of the standard specimen by base cycle count N0 in case of regular loading (MPa)	210
N0:	Base cycle count (S-N curve transition point)	107 = 1E+007
b1:	Slope of S-N curve first line	0.125
b2:	Slope of S-N curve second line	0.020

New dialog window appears.

7. Select the **Common properties** tab and set parameters as it is shown in the Figure 2.30.

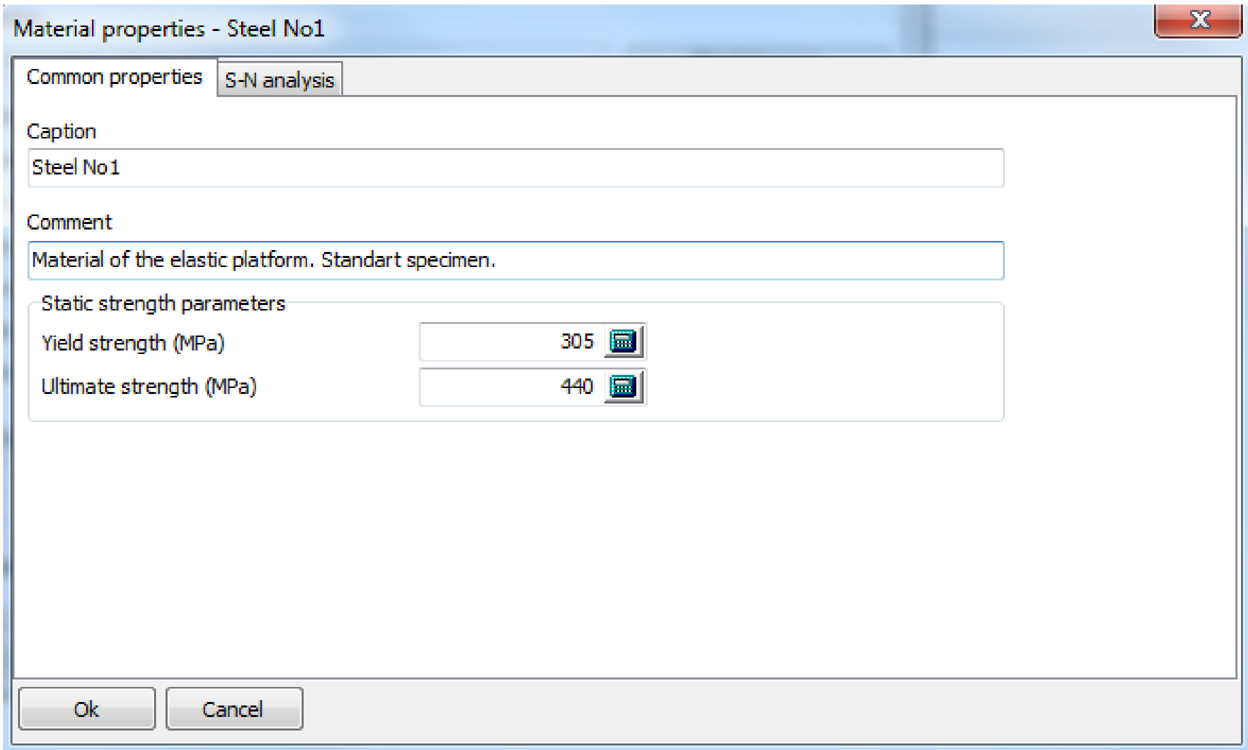


Figure 2.30. Material properties initialization: common properties

8. Select the **S-N analysis | Bend** tab to define fatigue resistance properties of the material.
9. Set **S-N curve** type to **Model No 5** and set material properties as it is shown in Figure 2.31.

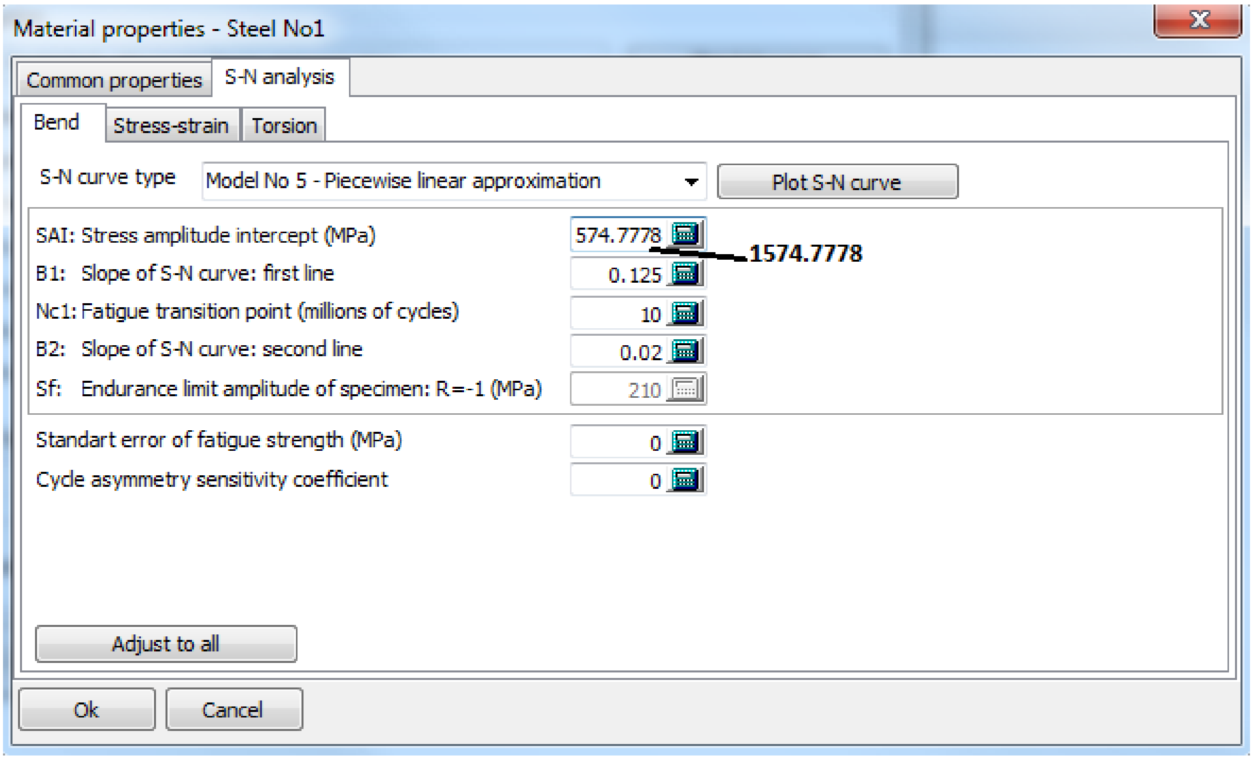


Figure 2.31. Material properties initialization: fatigue resistance

After initializing all properties click **Adjust to all** button.

10. Click **Ok** to save the new material and close the **Material properties** window. Material field of the **Control area properties [Top plate]** window will be defined with the new material caption.
11. Select **Bend** from the **Loading type**, see Figure 2.32.
12. Set **Coefficient of variation…** to **0.1**.

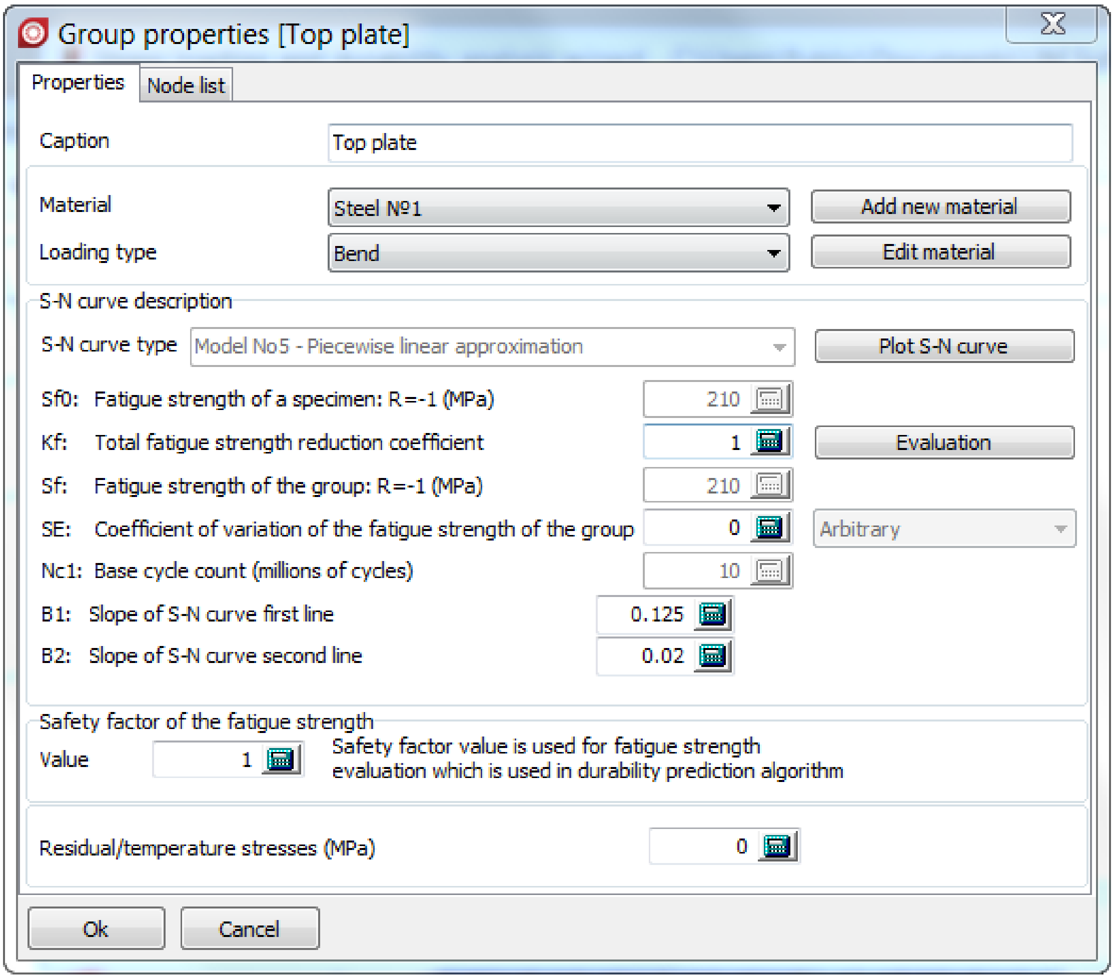


Figure 2.32. Group properties initialization: Top plate group

13. Click **Evaluation** to define **Total fatigue strength reduction coefficient** value, see Figure 2.33. Stress concentration factor is equal to **1** as soon as we use local stresses for this area. Click **Ok** when finish.

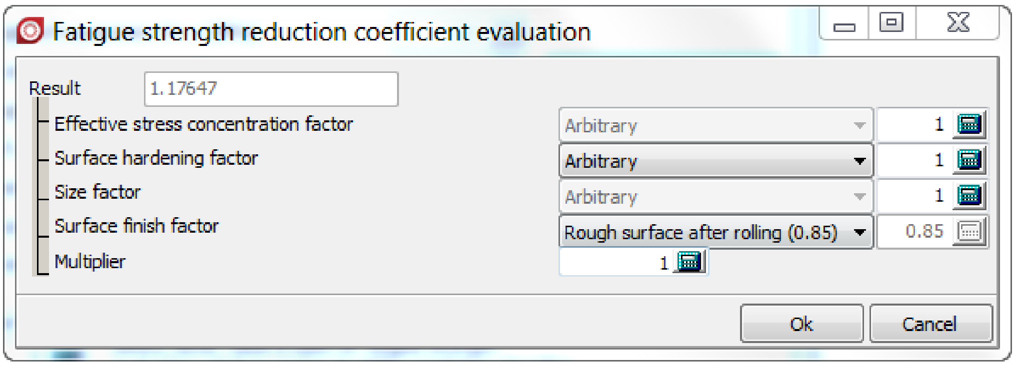


Figure 2.33. Total fatigue strength reduction factor evaluation form

14. Click **Ok** to save **Top plate** control area data, Figure 2.32.
15. Add one more control area and call it **Connections**, see Figure 2.34.
16. Define control areas properties as shown in Figure 2.34, Figure 2.35.

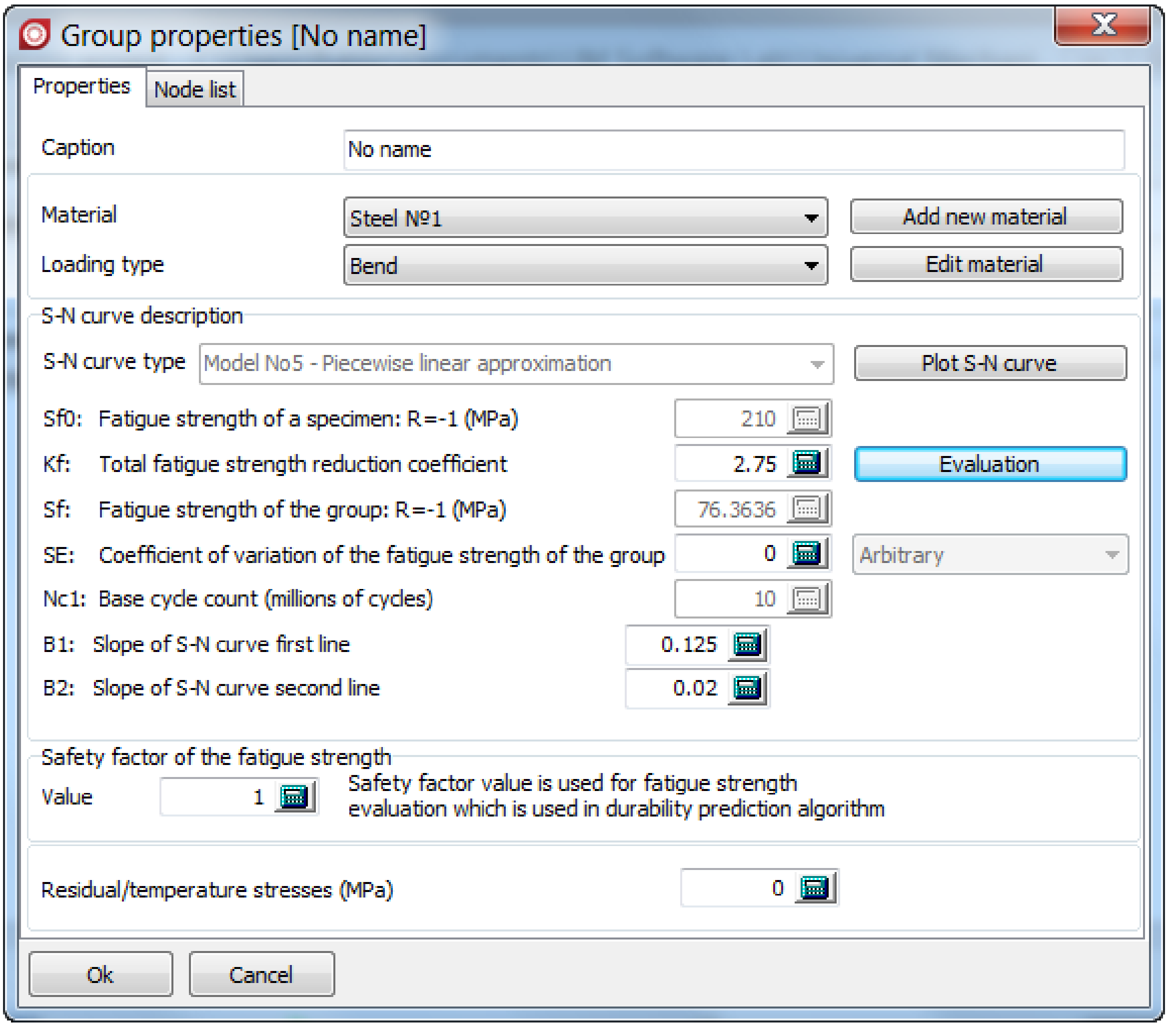


Figure 2.34. Initialization of group properties

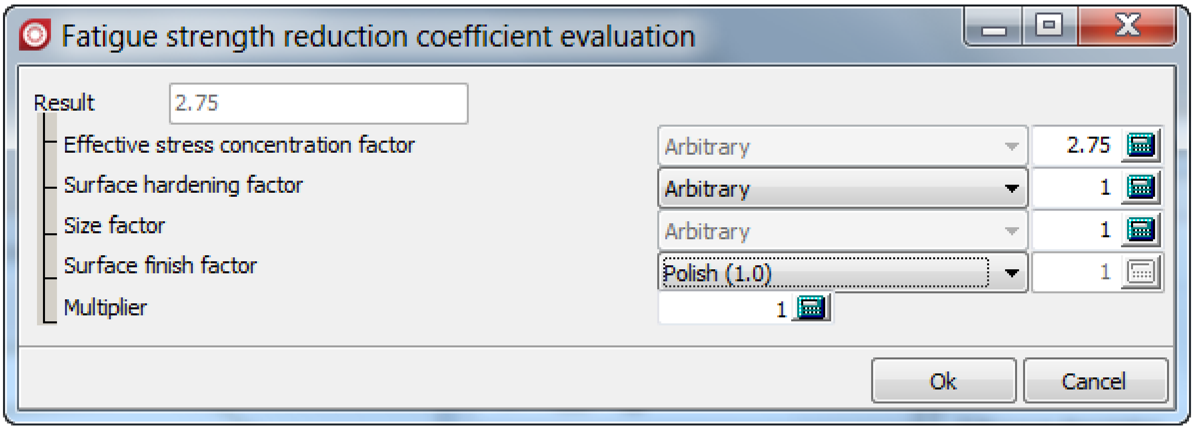


Figure 2.35. Fatigue strength reduction factor

17. Load a node list from the file *Connections node list.nls* using with the help of popup menu command **Load node list from file**, see Figure 2.36.
18. Click **Ok** to apply all changes and close the window.

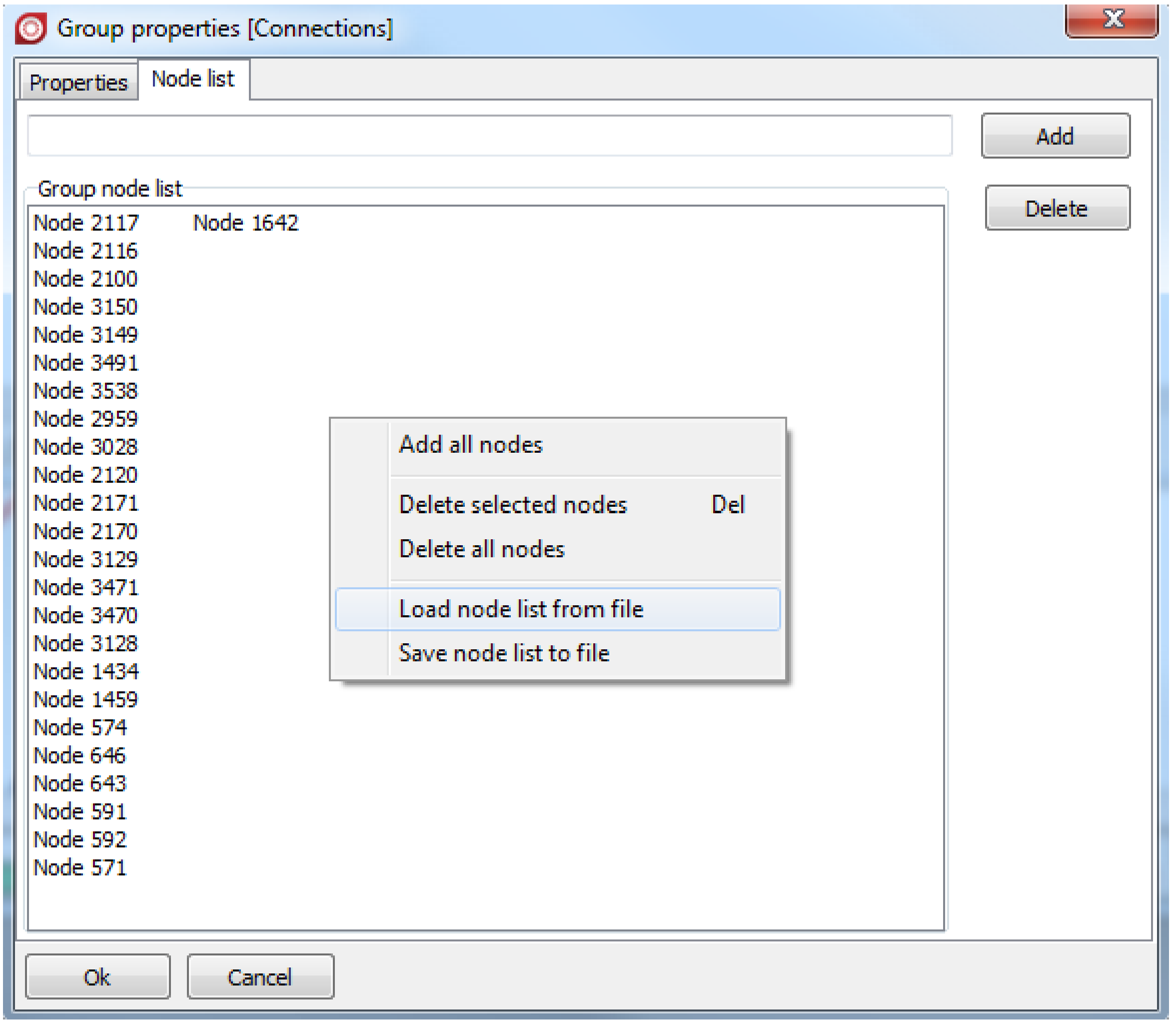


Figure 2.36. Load node list from text file

Fatigue resistance properties description is finished.

2.5.3. Save project to file

1. Use the  bottom in the top left corner of the **Stress loading and durability analysis wizard** to save project to file.

2.5.4. Durability parameters calculation

1. Select **Durability analysis | Evaluation** tab
2. Click **Calculate**, see Figure 2.37. It will take you less than a minute.

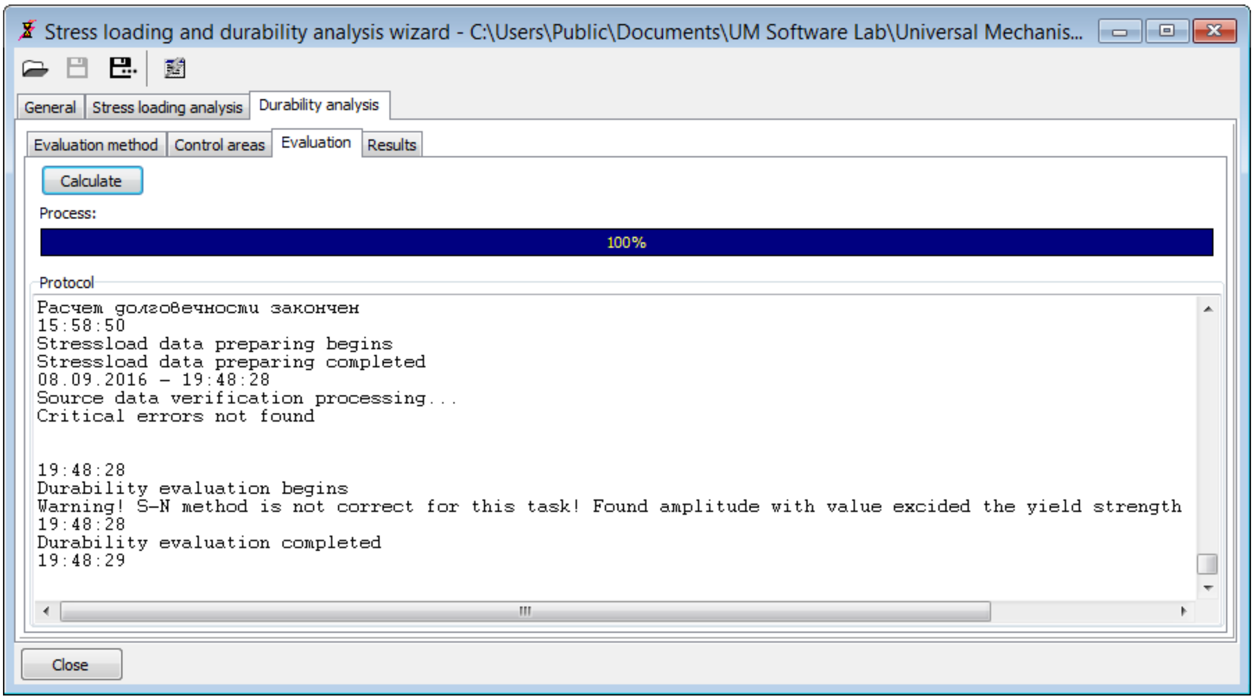


Figure 2.37. Durability parameters calculation tab

Now when durability analysis is executed we can come to analyzing its results.

2.5.5. Durability calculation results analysis

1. Select **Durability analysis | Results | Sensor list** tab.
2. Set **Load case** to **Combined stress load block**. Sort nodes by decreasing of *damage per a workday*, as it is shown in Figure 2.38.

Maximal damage is indicated at the node **592**. One can see that life-time of this area of the vibration table is the lowest and equals to **3930** workdays (**12.667** years of work). Accordingly, the fatigue strength of the construction is enough for the projected life-time (10 years).

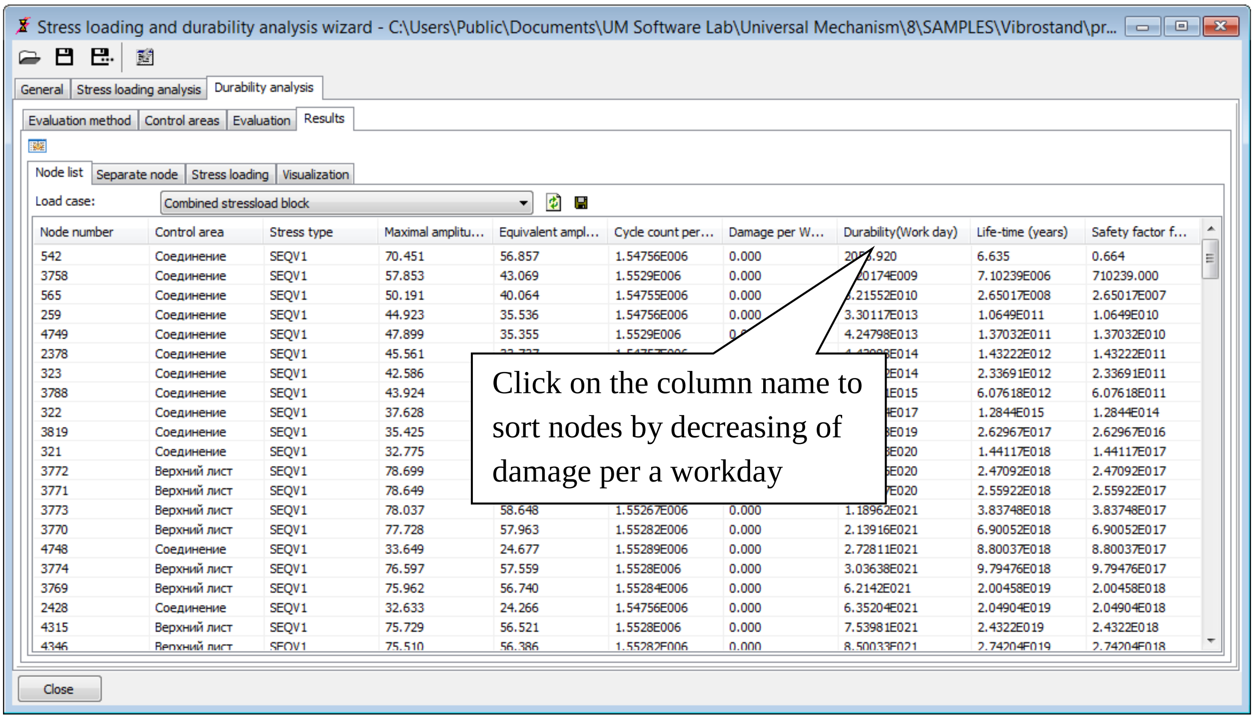


Figure 2.38. Durability results: sensor list

Let us see, which of the load cases can be preliminary called the most dangerous for this area.

3. Select **Durability analysis | Results | Single sensor** tab and select node number **592**, see Figure 2.39.

Maximal damage per workday corresponds to the **Stable work** load case.

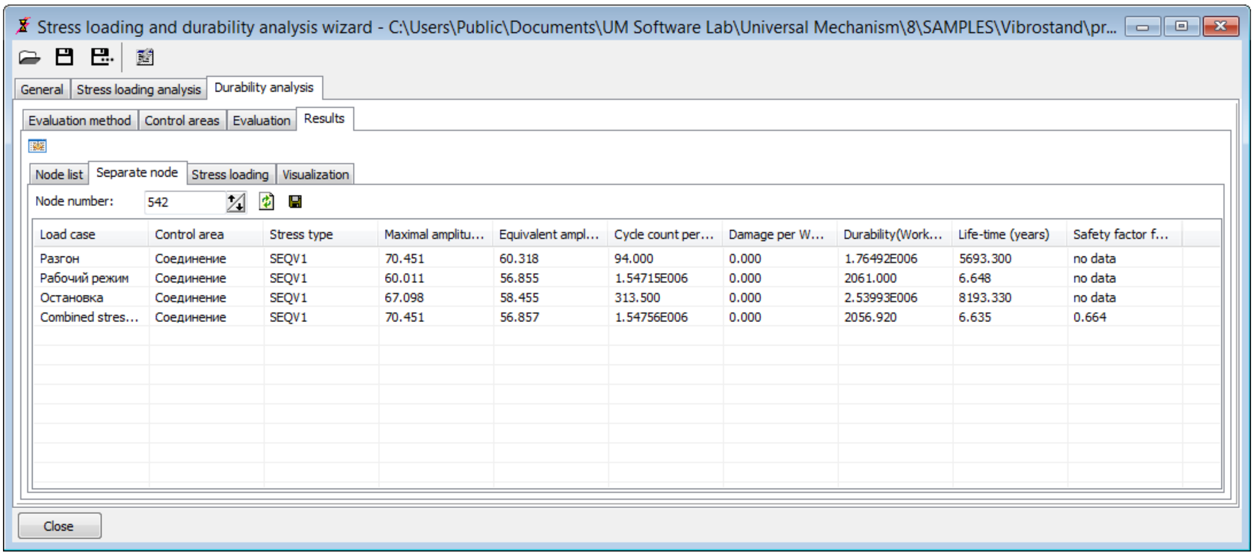


Figure 2.39. Results: single sensor

4. To inspect reduced amplitude distribution for the load cases at the node select **Durability analysis | Results | Stress loading** tab.
5. Input node number **592** and select the load cases to display distribution in table form, see Figure 2.40.

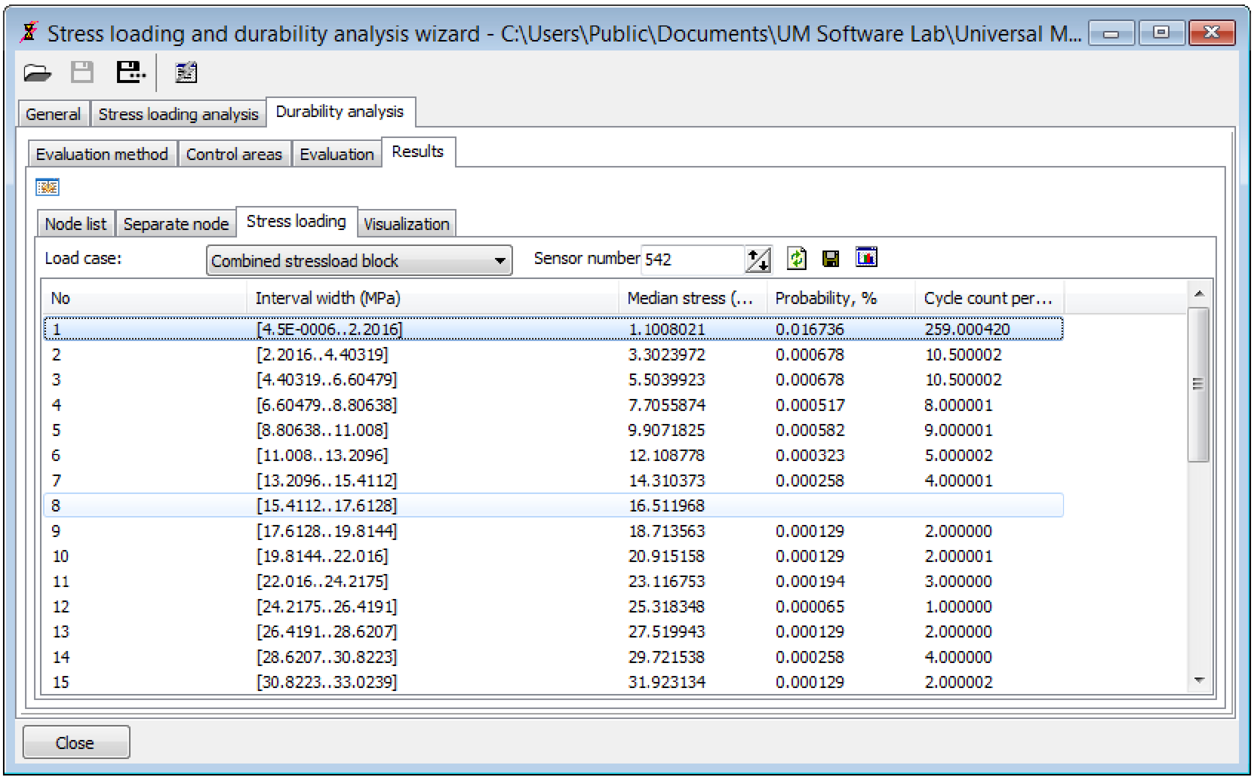


Figure 2.40. Results: stress loading

6. To plot the distributions click the  button. New graph window with plots of data appears, see Figure 2.41.

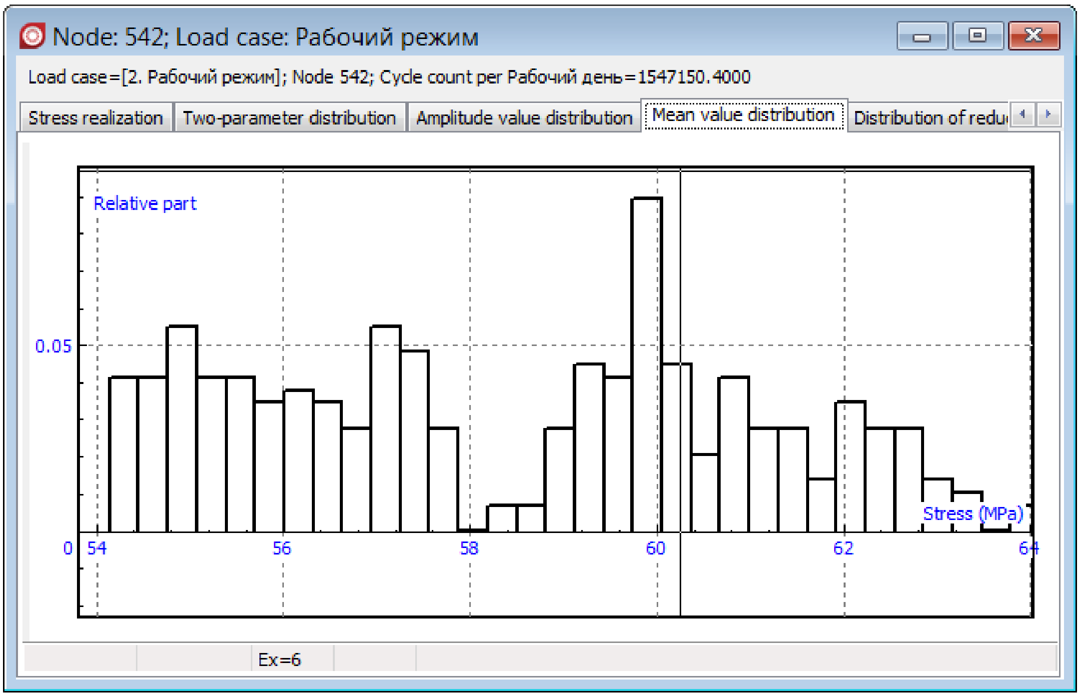


Figure 2.41. Reduced amplitudes distribution

2.5.6. Save project to file

1. Use the  bottom in the top left corner of the **Stress loading and durability analysis wizard** to save project to file.