STRENGTH ANALYSIS OF THIN-WALLED SPARS WITH WEB OPENINGS

Marek Horák
Brno University of Technology
Institute of Aerospace Engineering
Technicka 2896/2 616 69 Brno, Czech Republic
horak@fme.vutbr.cz

Abstract. This article provides information about a strength analysis of spars with thin web. Study, described in this paper, was carried out in order to investigate of the influence of web openings on critical web shear stress, distribution of transverse web deformations and von Mises stresses in such spars and also their ultimate load capacity. First of all, procedure for analytical calculation, based on available documents published by other researchers, was compiled. Secondly, one of nine specimens with different combination of web thickness and opening size were tested to failure. Thirdly, this specimen was analysed using finite element package MSC Patran/Nastran. Lastly, the results of experimental tests were compared with results of both analytical and numerical calculations.

Keywords. Strength analysis, spar, thin web, web opening.

1 Introduction

Beam with thin web is one of the basic structural elements in airframes of aircraft all types and sizes. In terms of load transfer, it is the most important structural element in the wing, empennage and control surfaces of small ultra-light and light sport aircraft. Its importance largely corresponds to the proportion of its weight on the weight of certain part of the aircraft. For example, in the wing with single spar structure, the weight of the beam is about 40% of the weight of whole wing. In comparison with the other fields of engineering, final weight of the aircraft structure is very importance. And just the weight reduction is precisely why aircraft engineers design thin-walled beams with cutouts in web. But in order to achieve maximum effect, it is necessary to use the latest scientific knowledge in this field.

1.1 Historical overview

Despite of its importance in aircraft structures, the issue of critical load, behaviour in post-critical stage of loading and ultimate load capacity of this type of beams remains somewhat neglected. Some important researches in this field were executed in the past. One of the most important is extensive research in American organization NACA (National Advisory Committee for Aeronautics) by Kuhn [1], [2], which was conducted during World War II and a few years later. Another important research was conducted by Timoshenko [3] in Russia at the same time. Their results are generally applicable theories, which arose in order to establish general dependencies and their description using empirical equations and graphs. Although these methods are conservative they were accepted and used by vast majority of aircraft producers at the time of its formation. Later, these researches were followed by another one, such as so-called Swedish research, which is also quite extensive. The results of several other studies, which were carried out several decades ago, can be found in some technical publications, which were written by for example Maddux [3], Niu [4], and other authors [5], [6]. Publicly available methods for design and calculation of spars with thin web with or without openings according to these methods has been further modified and supplemented by greater aircraft manufacturers in order to respect the latest knowledge and thus enable even more efficient use of the material. However, these modified forms have not been published for understandable reasons.

Modern approach for stress analysis of aircraft structures is application one of several available software packages, which works on the principle of finite element method (FEM). However, find correct solution by this way is also problematic because beam elements like web, stiffeners and flanges lose their stability. Necessary data and procedures, which lead to correct solution, are also not disclosed. In addition, generalize the numerical solutions for different variants of structure are difficult.

1.2 Study objectives

Currently there is not publicly available comprehensive and experimentally validated method (or manual) for calculation of the spars with a thin web with openings, including the latest knowledge in this field. This is why the small aircraft producers, which are unable to perform their own costly studies, are addicted to the results of several decades old researches.

The main objective of this study is verification, modification and supplementation of the available calculation methods for determining the ultimate load capacity of the spars with thin web with or without circular web openings, which are commonly used in design of small airplanes. Emphasis is also placed on assessment of critical web shear stress and influence of diagonal tension field on stress in stringers, flanges and rivet joints.

2 Experimental investigation

The initial phase of this study was focused on determining the fundamental parameters of the problem. Specifically, basic shape, overall size and several another important dimensions of the specimens were determined. After that, rigid construction of a test jig (frame and load distribution system) was designed in accordance with the desired way of specimen mounting and application of load. It was done in order to achieve a maximal compliance with the conditions in real aircraft structures.

The purpose of the experimental investigation is verification and validation of the results of the currently available computational methods, their modification, and also obtaining a necessary data for the preparation of an adequate numerical model.

Although all nine specimens were designed and produced only one of them has been experimentally investigated yet.

2.1 Description of the specimen

As previously mentioned, nine specimens were designed and produced. The dimensions of all specimens (spars) are identical in all cases, but their structure is slightly different. All nine spars consist of web either with or without cutouts, two flanges, nine stiffeners, four hinges and the same number of doublers. Structure of specimen 1C is shown in Figure 1.

The total quantity of specimens is divided into three series. Each series consists of three spars, which have the same web thickness, but one spar has web without openings, another one has openings with a small diameter and last one has openings with a large diameter. Basic geometrical characteristics of the specimens are presented in Table. 1.

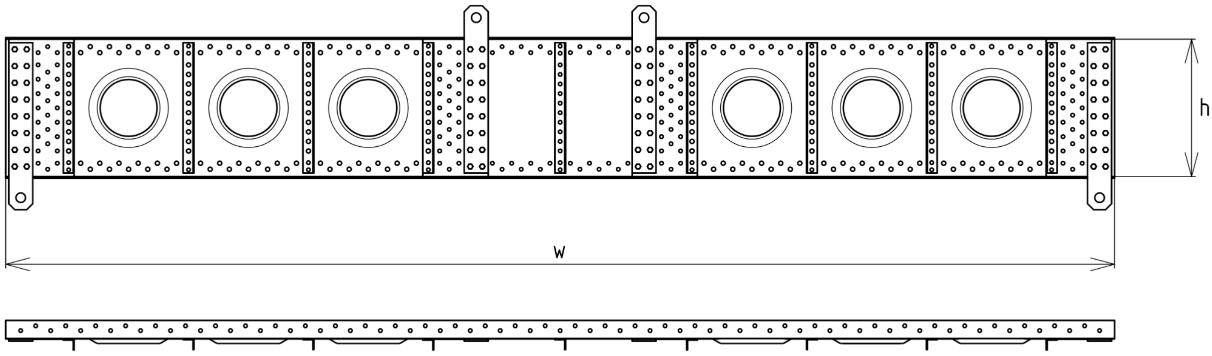

Figure 1: Typical specimen structure.

Geometry of openings corresponds to Czech aviation standard LDN 1505. Next difference between specimens is in geometry of flanges. Specimens with lower assumed ultimate load capacity have flanges formed by one L-shape bar with dimensions of 30x3 mm and specimens with higher assumed ultimate load capacity have flanges formed by two L-shape bars with the same dimensions and thin sheet strip. In all cases, single sided stringers are formed as bar with L type cross section and dimensions of 18x20x1.2x2. Doublers are used only for web reinforcement on the edges of the areas which are examined.

Specimen ID	*Specimen width*	*Specimen height*	*Web thickness*	*Opening diameter*	*Flanges area*	*Effective height*	*Distance between stiffeners*	*Parameters*	
	w [mm]	h [mm]	t_w [mm]	D [mm]	A_F [mm^2]	h_e [mm]	d [mm]	D/h_e [-]	h_e/t_w [-]
1A	1850	230	0.6	-	207.5	211.3	200	-	351
1B	1850	230	0.6	63.5	145.0	210.4	200	0.30	351
1C	1850	230	0.6	96.0	145.0	210.4	200	0.45	351
2A	1850	230	0.8	-	290.0	210.4	200	-	263
2B	1850	230	0.8	63.5	182.5	211.1	200	0.30	263
2C	1850	230	0.8	96.0	145.0	210.4	200	0.45	263
3A	1850	230	1.0	-	390.0	211.2	200	-	211
3B	1850	230	1.0	63.5	290.0	210.4	200	0.30	211
3C	1850	230	1.0	96.0	182.5	211.1	200	0.45	211

Table 1: Geometrical characteristics of specimens.

Webs and doublers were made from thick sheets (material is Z424253.61 aluminum alloy), flanges and stiffeners were made from extruded bars (material is Z424203.61 aluminum alloy). Hinges were made from steel plate. The connections between spars and hinges are realized using steel bolts with diameter of 5 mm, whereas solid aluminum rivets with diameter of 3.5 mm were used in all another connections.

2.2 Description of the test jig

The main purpose of the test jig is to ensure the correct boundary conditions and simultaneously correct application of load to the structure of all nine specimens. Structure of the test jig is shown in Figure 2. The specimen is attached to the test jig at both ends by hinges. On one side of the specimen, only rotation about the pivot axis, which is perpendicular to the plane of the web, is allowed. On the other side of the specimen, the same rotation and also horizontal displacement in the plane of the web are allowed. This way of attachment ensures a natural deformation of the specimens during tests. Furthermore, to ensure the boundary conditions approximately corresponding to the real wing structures, four guiding frames are used. These frames are used for preventing unwanted rotation and out of web plane displacement of the cross section in places, where the specimens are attached to the test jig and also in places, where the specimens are connected to the load distribution system.

For purpose of loading, only one hydraulic actuator, which is connected to the load distribution system, is used. Load distribution system is connected to two inner hinges and distributes the total load evenly to both symmetrical half of the specimens (see Figure 3). This test jig is used for testing of all nine specimens.

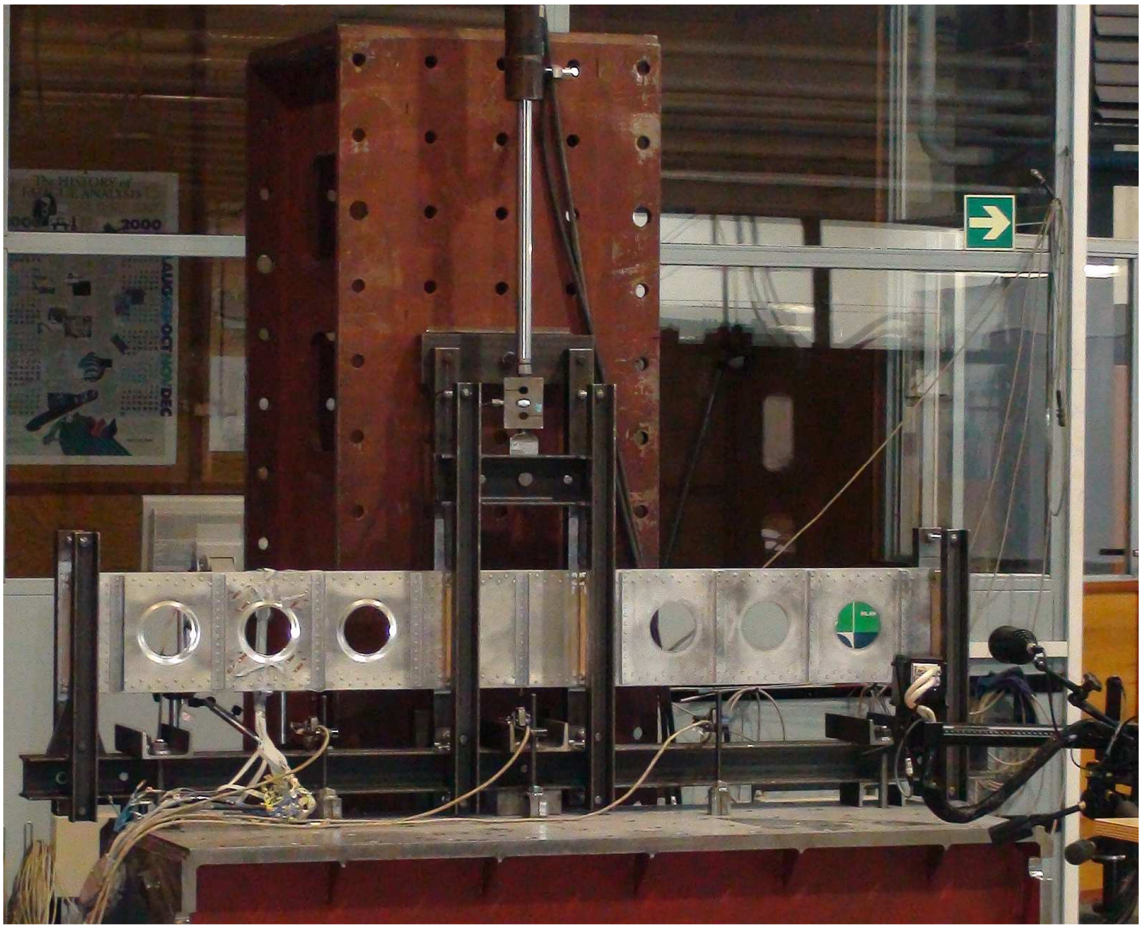

Figure 2: Specimen 3C in the test jig before testing.

2.3 Measurement

Preparation of the specimen and test rig was carried out and also measuring systems were installed and calibrated before each measurement.

During tests, total twenty seven strain gauges and five linear variable displacement transducers were installed on specimen to measure strains and displacements in defined positions. Also a photogrammetric system Aramis was used for measuring strains in web. All strain gauges were placed nearby a central web field on one half of the specimen, while a system Aramis was used for measuring strains in three web fields on second half of the specimen. Applied load was measured between hydraulic actuator and load distribution system (see Figure 3).

Data from load gauge, strain gauges and linear variable displacement transducers was recorded using data acquisition system ESAM CF.

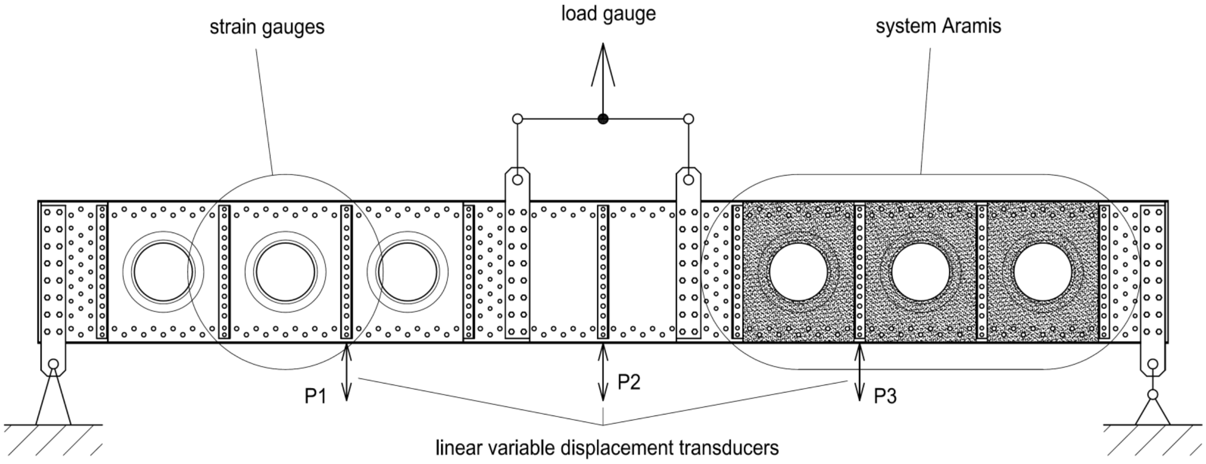

Figure 3: Locations of points and areas for measuring strains and displacements.

Sixteen strain gauges were placed on the web, eight at the front and eight at the back of the specimen. These strain gauges were installed for purpose of determination of influence of opening on diagonal tension field in buckled web during post-critical stage of loading. Next four strain gauges were placed on upper flange, the same number of strain gauges was placed on lower flange and three strain gauges were placed on one stiffener. These eleven strain gauges measured additional loads, which were caused by diagonal tension field. Position and marking of strain gauges is shown in Figure 4. Strain gauges T1B … T8B are installed on the same places as strain gauges T1A .. T8A but they are installed on the other side of the web.

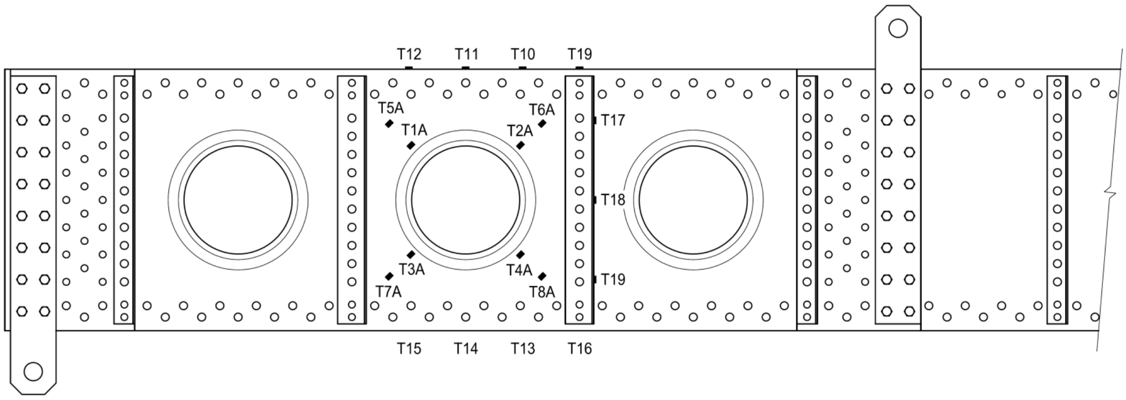

Figure 4: Locations and marking of strain gauges.

During tests, these four parameters were measured:

- deflection curve of the spar
- stress in flanges and stringer
- stress in web (near opening)
- transverse deformation of buckled web

3 Finite element analyses

The purpose of the numerical simulations is finding and compilation such computational model, which will lead to almost the same results as in case of analytical calculation and experimental investigation. For finite element analysis MSC software package Patran/Nastran was used.

3.1 Finite element model

Spars were modeled using 1D beam and 2D shell elements only. Geometry of each model corresponds to the real geometry of the produced specimens. Components such as web, doublers, flanges, striffeners and hinges were represented using 2D shell elements, while all connections (rivet and screw joints) were represented using 1D beam elements.

The Isomesh with element size of 3 mm was used for meshing the flanges and the stiffeners. For meshing the web with or without openings and doublers, booth Isomesh and Paver type of mesh with various element size were used. Hinges were modeled as combination of Isomesh and Paver mesh. Each element had defined its real properties, such as thickness in case of 2D element or diameter in case of 1D element. Finite element model of a specimen 3C is shown in Figure 5.

Two types of materials were implemented in the model. Elasto-plastic materials, which were defined using true stress-strain curve of the materials, were used for all aluminium parts (web, doublers, flanges, stiffeners and rivets). Linear-elastic material with correct properties was used for steel hinges and screw joints.

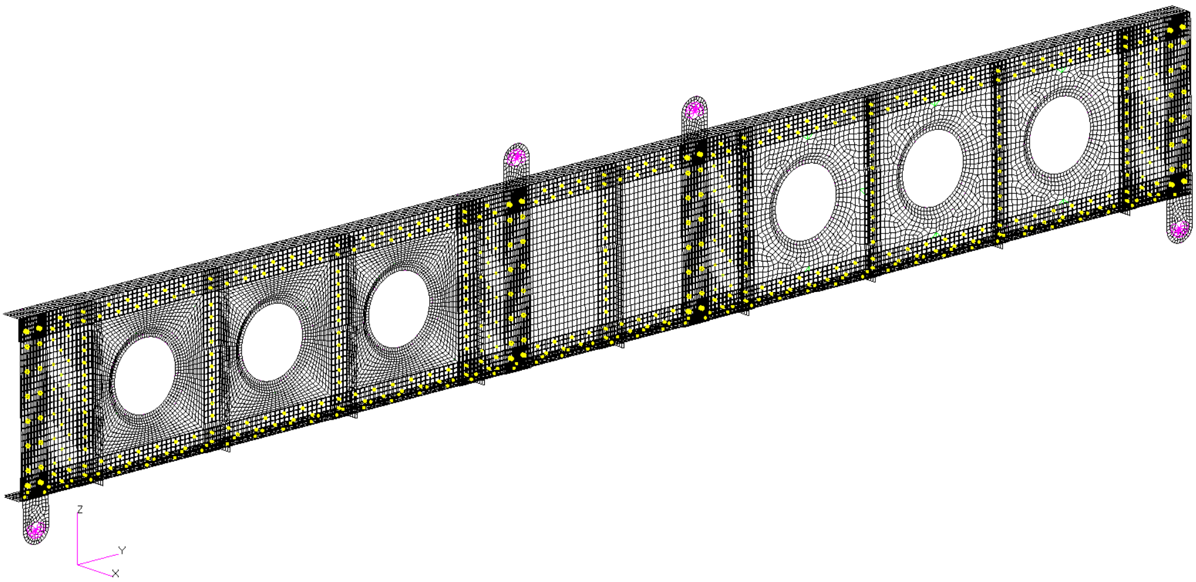

Figure 5: Finite element model of specimen 3C.

3.2 Analysis parameters

In numerical simulations, mainly the following factors are investigated:

- size of initial geometric imperfection of the web
- type of mesh (only web mesh)
- size of elements (primarily web elements)
- contacts between adjacent surfaces

3.3 Loading and boundary conditions

Boundary conditions, which were applied in numerical model, were corresponded to practical experiments (see chapter 2.2 for details). Outer hinges on one side of the specimen allowed only rotation around a pivot axis, outer hinge on the other side of the specimen allowed rotation around a pivot axis and also its displacement along span of the spar. Both outer and inner hinges were prevented from out of web plane moving. Total load was divided into two equal forces, which are applied to the inner hinges.

Also an application of contacts was considered. However, contacts had only negligible effect on the results and computing time was significantly higher in comparrison with the same job without contacts. This is the reason why the contacts were not used in further analysis.

4 Results

In this chapter, the results of experimental investigation performed on specimen 3C and the results of numerical calculation of the same specimen are presented. Both experimental tests and numerical analyses of remaining eight specimens have not been performed yet. Therefore generally valid conclusion is not contained in this chapter. The content of next text is primarily a conformity assessment of developed numerical model with the real behavior of the specimen during tests and also a comparison of these results with theoretical assumptions.

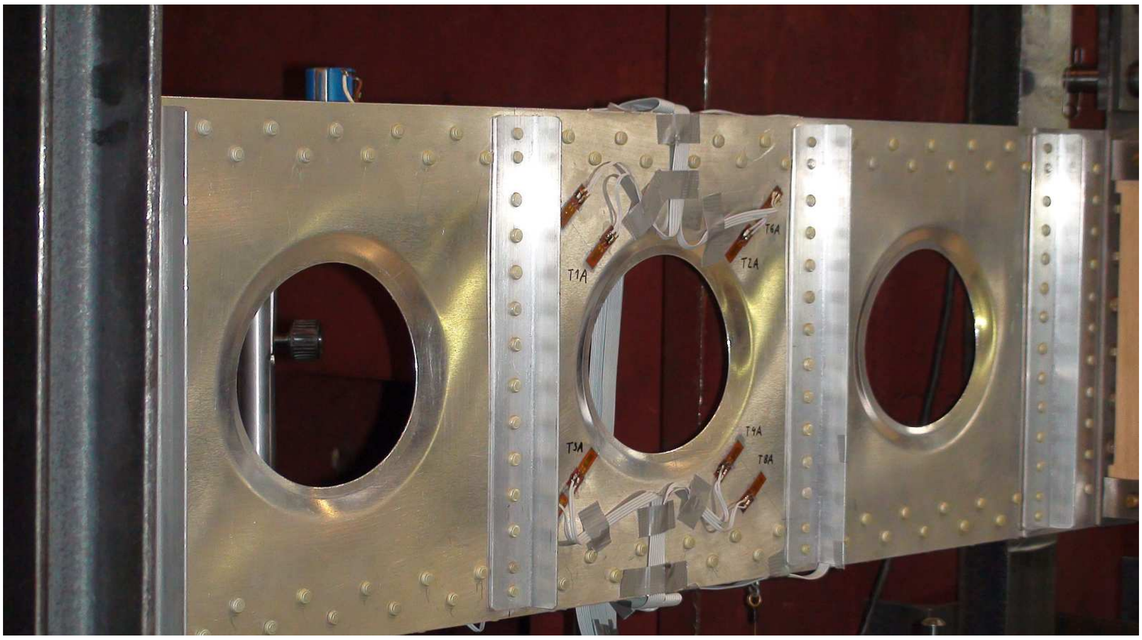

Figure 6: Bucked web during test.

In Figure 6, there is a specimen 3C during test. This figure clearly shows a buckled web during post-critical stage of loading. When the load exceeds critical web shear stress, a diagonal tension field beam will be created. Diagonal tension field is associated with additional stress in flanges, stiffeners and rivet joints.

Because this specimen will be subjected to further tests, the total load during performed tests did not exceed an expected minimal value of ultimate load capacity. Minimal expected value means the lowest ultimate load capacity of values, which were calculated using several available methods [1] [3] [4]

[6]. Anyway, each of them predicts different value with a difference in the order of one hundred percent! It has been shown experimentally, that the value of critical web shear stress was corresponded to the expected value. This value was determined on the basis of available procedures and converted to value of total load. All methods, which were used for calculating critical web shear stress, lead to almost the same value of critical load.

Deflection of the specimen in measured sections (positions of linear variable displacement transducers are shown in Figure 3) and the Maximal principle strains in the central field of the web (positions of strain gauges are shown in Figure 4) depending on total load are shown in Figure 7.

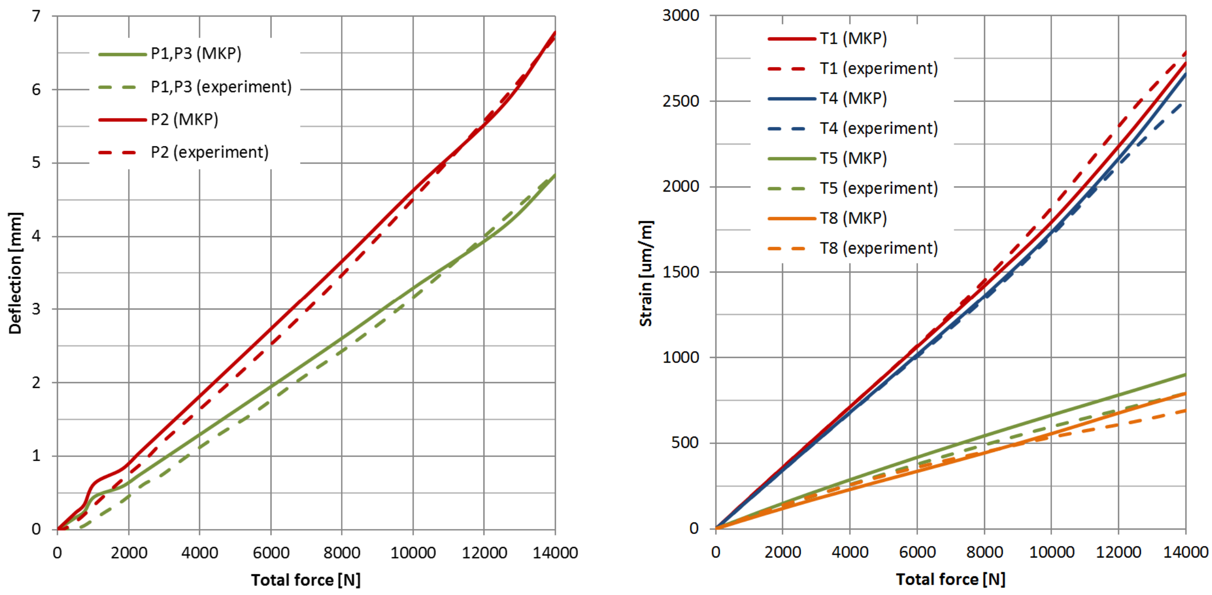

Figure 7: Comparison of measured and calculated (MKP) deflections and strains.

In these diagrams, the parameters determined on the basis of numerical calculation are shown also. A correlation between the results of numerical simulation and practical experiment is obvious. In the diagram on the right, an influence of diagonal tension field is slightly evident. A slope of the curves is changing, when total load is equal to critical web shear total force, i.e. when it is equal to 9 000 N. This phenomenon is very good visible in the case of strain gauges T1 and T4, which are located close to the opening.

Figure 8 and Figure 9 show the stress distribution on the web surface at post-critical load, when the total load is equal to 14 000 N. The stresses, which were measured during experiment using the system Aramis are shown on the left side, whereas stresses, which were calculated using finite element method are shown on the right side of these figures. Booth results are nearly identical. This suggests that this method of measuring is suitable for this type of instability problems.

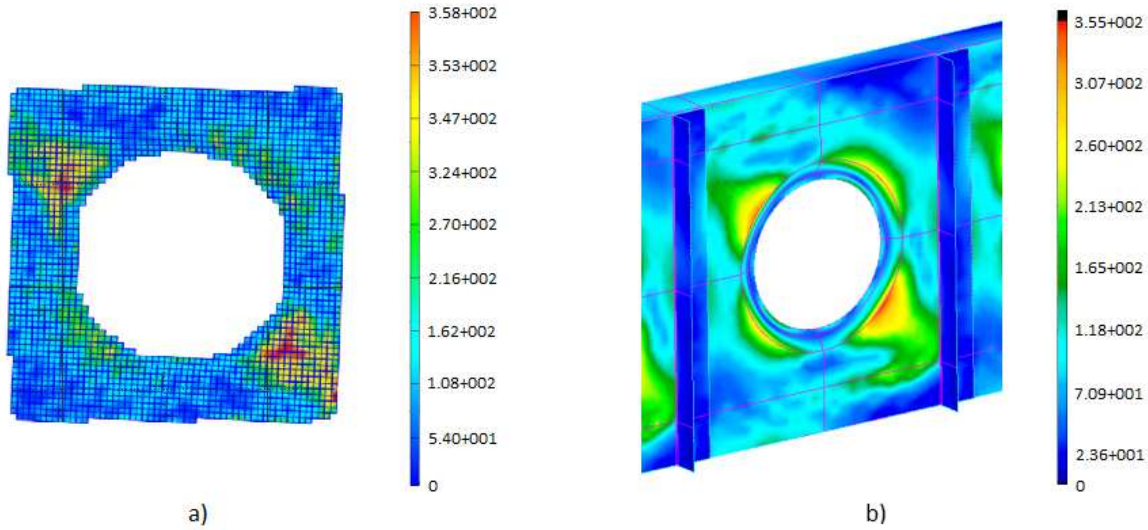

Figure 8: Von Mises stress [MPa] in central field of the buckled web
from: a) system Aramis, b) numerical simulation.

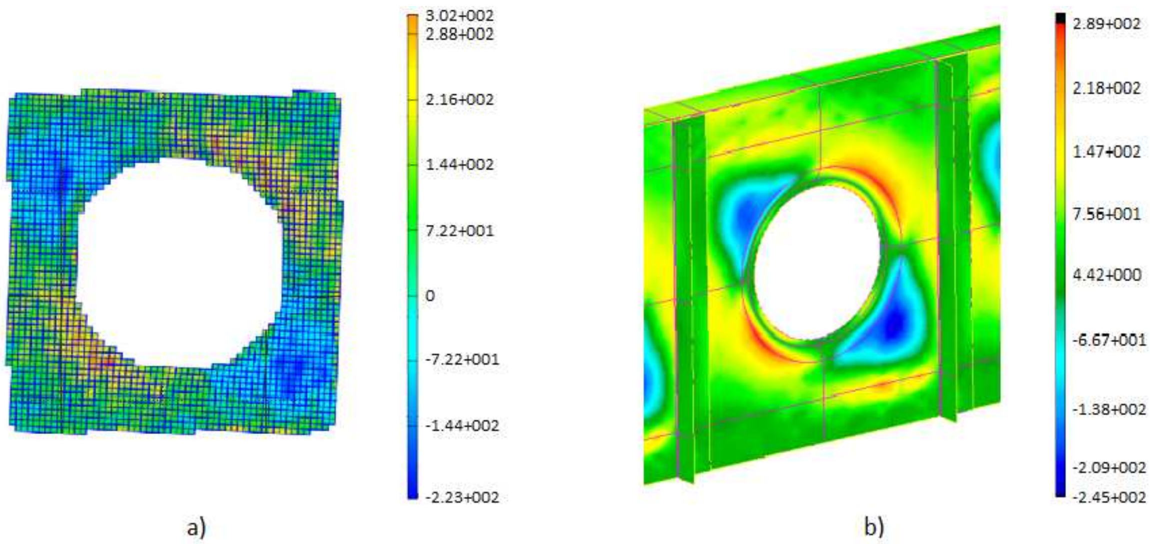

Figure 9: Maximal principal stress [MPa] in central field of the buckled web
from: a) system Aramis, b) numerical simulation.

It has been proved, that system Aramis is suitable for measuring such parts of structure, which has sufficiently large and smooth surface without abrupt changes in geometry. This system is not suitable for measuring stresses in flanges and stiffeners, because large number of rivets prevents proper recording of the data and its evaluation. In these structural elements, the measuring using strain gauges have been proved. Measuring stresses in web using strain gauges is somewhat problematic as well. This type of measurement is reliable, when the applied load is lover then the critical load. The problem occurs, when applied load is higher than its critical value. This stage of loading is always associated with a significant deformation of the web. And just large bending deformation, which is unevenly distributed across the surface of strain gauge, can bring significant inaccuracies into measurement.

5 Conclusion

This study is quite extensive and it is currently solved. The remaining specimens will be prepared for testing soon. As well as methodology for the numerical solution of these types of structures will be completed in a several months. Methodology for calculation of spars with thin web with or without openings, which was mentioned above, its description and the description of the process of its composition are very extensive issues and they are out of frame of this article.

In the future, an extension of this study is planned. This extension will be focused on compilation of a process leading to optionally designed structure and creation software, which will greatly accelerate the process of designing this type of structural elements.

Finally, there is one note about a possibility of use of this study. It is expected that the results, mainly the methodology for calculation, will be used in design stage of the production of a small metal aircraft.

References

[1] Kuhn, P., Peterson, P., Levin, L.: A Summary of Diagonal Tension, Part I – Methods of Analysis, Langley Aeronautical Laboratory, Washington, 1952, Report No. TN-2661

[2] Kuhn, P.: The Strength and Stiffness of Shear Web With Round Lightening Holes Having 45° Flanges, Langley Memorial Aeronautics Laboratory, Washington, 1942, Report No. WR L-323

[3] Maddux, G., Vorst, L., Giessler, F., Moritz, T.: Stress Analysis Manual, Technology Incorporated, Dayton, 1969, Report No. TI-219-69-24

[4] Niu, M. C.: Airframe Stress Analysis and Sizing, Second Edition, Conmilit Press Ltd., Hong Kong, 1999, ISBN 962-7128-08-2

[5] Singer, J., Arbocz, J., Weller, T.: Buckling Experiments: Experimental Methods of Buckling of Thin-Walled Structures, Volume 2, John Wiley & Sons, Inc., New York, 2002, ISBN 0-471-97450-1

[6] Čtverák, J., Mertl, V., Píštěk, A.: Soubor podkladů pro pevnostní výpočty leteckých konstrukcí, LÚ FSI VUT Brno, Brno, 1997