Simulation of stress in an innovative combination of composite with metal sheet

A Wróbel¹, M Płaczek², A Buchacz³, A Słomiany⁴
¹,²,³,⁴ Silesian University of Technology, Faculty of Mechanical Engineering,
Institute of Engineering Processes Automation and Integrated Manufacturing Systems,
Konarskiego 18A, 44-100 Gliwice, Poland

E-mail: Andrzej.wrobel@polsl.pl

Abstract. In this article research of stress impact in multi-point connection glass epoxy composite with a metal sheet with a rivet nuts was presented. Composite materials are increasingly used because of the good mechanical properties and low price. The laminates are composites of a layer structure, characterized by very high strength in the direction of the fibers, their weakness is not good toughness in a direction perpendicular to the layers. Mainly checking of displacements and stresses generated on the sheet as a result of pneumatic actuators load for composite boards was carried out. Glass-epoxy composite consisting of four layers of glass mat with a weight of 1000 g/m² and an epoxy resin and hardener HG700 LG700 volume ratio of 38/100 was created. Next composite was fixed with steel plate with a rivet nuts and bolts. A model of laminate samples and plate was simulate in Siemens NX 8.5 software. The simulation results will determine stresses and displacements in conjunction newly designed composite sheet. Strength analysis was performed with the use of the module NX Advanced Simulation. FEM is an advanced method for solving systems of differential equations, based on the division of the field into finite elements for which the solution is approximated by specific functions, and performing the actual calculations only for nodes of this division. Due to the complexity of the created object to simplify the elements made to reduce the calculation time. This article presents the study of stresses and displacements in the composite plates joined with sheet metal, in summary of this article, the authors compare the obtained results with the computer simulation results in the article: "The study of fix composite panel and steel plates on testing stand".

1. Introduction
At the Faculty of Mechanical Engineering Technical University of Silesia are conducted research on the analysis and synthesis of mechanical systems and mechatronics [3,12,13,16], concern both theoretical [1,2] considerations how practical industrial tasks [14]. Recent works undertaken relate to research the properties of composite materials [7,9] as components and assemblies railway wagons [5,8,11]. Some of the work is conducted in collaboration with other research centers [10]. The aim of work is to study the impact of multi-emphasis pneumatic actuator for connection glass-epoxy composite material with a metal sheet using rivet nuts. Especially important was to focus on checking the displacements and the stresses caused to metal sheet as a result of pressure of cylinders on the composite material.
The width of the gap emerged during the test was examined. The displacements were measured by mechanical dial gauge, tension was measured using resistance extensometer and the gap using feeler gauges. Glass-epoxy composite was created which consisted of 4 layers of glass mat with a weight of 1000 g/m² and the epoxy resin HG700 and hardener HG700 with volume ratio of 40/10. Then the composite material was combined with metal sheet S235JR having a thickness of 4 mm by means of blind rivet nuts and bolts M8. Models and samples were made of laminate and metal sheets, conducted simulations MES in software SIEMENS NX 8.5. The simulation allowed to determine unfavorable resultant displacements and stresses for the connection, and the resultant width of the gap. Models and simulations were made for five samples.

2. Measurements Simulation stress in the composite plates
The Siemens NX program is a modern and integrated CAx system, which offers a rich set of software tools to support the entire series of activities related to the process of construction and manufacturing of the product.

The main feature that distinguishes the NX system from other systems supporting the design is a comprehensive possibility of technological operations during all stages of production. During the study, following modules were used:

- Modelling;
- Assemblies;
- Advanced Simulation.

The first one is the basic application Siemens NX software. It allows you to create complex parts including both solid modelling and profiles created in the sketches or imported from other CAD programs. Assemblies application supports the creation of clusters of parts. It enables the creation of complex assemblies with the maintenance of the common bonds between the individual elements. Assembly elements can be placed at the relation to local or global reference systems. Advanced Simulation Module is an application type CAE. The system has built-in database of basic materials, which can be extended by formulating their own substrates and materials. The system allows to use the built-in solvers, of which the most popular is the NX Nastran. After finishing the analysis, the results are available in clear graphic form, providing the interpretation of the results.

MES is one of the basic tools of computer-aided research and engineering analysis of a very extensive range of use and high popularity. This method is based on the adoption of the approximation of the displacement field and stress field or a combination of these approximations in each element.

Troublesomeness of process of stress analysis of the projected structure was minimized because of the knowledge and experience of engineers characterising the border conditions and the method of weighing the construction

3. Research plan
Before attempting to simulation of connecting composite material with metal sheet with help of rivet nuts, a plan of tests was written according to the following points. Points in bold font refer to this article.

- Making a four-layer composite.
- The cut of metal sheet into the proper size.
- Execution of riveted connection on steel sheet S235JR.
- The combination of composite and steel sheet by means of screws M8, which mounted in blind rivet nut.
- **Modeling of samples in Siemens NX.**
  - Modeling of the first sample, ie. The same metal sheet.
  - Modeling of the second sample, ie. Sheet with holes.
  - Modeling third sample, ie. Sheet with blind rivet nuts.
  - Modeling the fourth sample, ie. Sheet with rivets bolted to the glass-epoxy composite.
Execution of finite element method FEM simulation on modeled samples.  
Analysis the results of FEM simulation.  
  o Displacements analysis on the samples.  
  o Stress analysis on the samples.  
  o Analysis of the width of the gap in combining metal with composite material.  
  o Stress analysis on individual layers of the composite.  
Comparison the results of computer simulation with the actual test results. 

Modelling the metal sheet and the composite material was carried out in the Modelling module. To create the model, the BLOCKoption was used, from which shape of metal sheet and composite material was given. The next step was to create holes in the metal sheet and composite plate. The holes were created by using the PRE-NX5 HOLE function.

![Figure 1. BLOCK window](image1)

![Figure 2. PRE-NX5 HOLE window](image2)

Created connections of metal sheet with composite material. For this purpose ASSEMBLY function was used. Using the ADD COMPONENT window both components were added and combined in bonds with ASSEMBLY CONSTARINS tool, bonds type of TOUCH and ALIGN were used.
After completion of the modelling of connection of composite material with metal sheet strength calculations began using the finite element method. For the calculation of the module Advanced Simulation was used.

Structural analysis and the type of NX NASTRAN solver were chosen. Grid to simulate 3D Swept Mesh was selected. It is a grid composed of squares of Chex elements in size of 5mm.

The type of sheet material was selected and material parameters were entered into NX. After completion of all steps metal sheet grid (Picture 4) was received. Then composite material mesh was modelled and the Create Physical option was selected, which allows to open laminate modelling window. After creating a Solid Laminate Modeler window, it is possible to create any laminate based on layers.

In the bookmark Ply Layup Create New Ply icon was selected to create 1 layer of the laminate. After pressing the select ply material icon, it is possible to select a new material which is composite fibers. To choose this option one should select the type of the fiber in the Ply Laminate Material
Manager window. Selecting a matrix material which is an epoxy resin in the matrix volume fraction 0.5 and reinforcement of the composite material.

![Figure 6. Isotropic Material window.](image1)

![Figure 7. Solid Laminate Modeler window with layers composite.](image2)

After creating all the properties of the fibers in the Solid Laminate Modeler window, 4 layers of the composite material are created. The thickness of the individual layers equals 0.9 [mm]. To get the results of stress analysis Stress or Strain Output Request option needs to be selected. After defining the contact, model is ready for simulation of finite element FEM.

![Figure 8. Modelled combination of composite and metal sheet.](image3)

![Figure 9. Displacement of connecting the metal sheet with composite material.](image4)

Number of computer simulations and analysis was performed and this article contains only demonstrative result. In the first simulation research was performed on metal sheet with drilled holes. On Figure 8 the model with imposed mesh was presented. The material chosen for the metal sheet is steel with the similar properties to the steel S235JR.

The values of the measured displacements were shown in Table 1.
Table 1. Displacement values.

| Pressure values in cylinders [Bar] | Maximum values [mm] | Medium values [mm] | Minimum values [mm] |
|-----------------------------------|---------------------|--------------------|---------------------|
| 6                                 | 1.779               | 1.776              | 1.772               |
| 4                                 | 1.186               | 1.185              | 1.182               |
| 2                                 | 0.593               | 0.592              | 0.591               |

The next step in the simulation was to study stress in different places on metal sheet. The values of stress were presented in Table 2.

Table 2. Stress values of the composite with metal sheet model.

| The pressure values [Bar] | Stress values | Extensometer 1 [MPa] | Extensometer 2 [MPa] | Extensometer 3 [MPa] |
|---------------------------|---------------|----------------------|----------------------|----------------------|
| 6                         | Maximum       | 56.813               | 80.415               | 33.642               |
|                           | Medium        | 55.860               | 77.215               | 31.726               |
|                           | Minimum       | 54.963               | 73.794               | 29.286               |
| 4                         | Maximum       | 37.223               | 53.904               | 25.796               |
|                           | Medium        | 36.734               | 52.305               | 25.052               |
|                           | Minimum       | 36.089               | 50.475               | 24.220               |
| 2                         | Maximum       | 18.896               | 26.958               | 12.791               |
|                           | Medium        | 18.337               | 26.184               | 10.866               |
|                           | Minimum       | 17.955               | 25.246               | 9.566                |

The last step was to measure the width of gaps formed by connecting composite material with the metal sheet, the gaps were formed on the top and bottom connections. Width of the gap was measured for all load cases of a model, the values were presented in Table 3. To measure the gap annotation option was used, then the two edges of the nodes on the model were indicated, the lower edge of the composite node and the top edge of the sheet node. On the received annotations subtraction operation was performed which allowed get the width result.

Table 3. The values of the width of gaps.

| The pressure values [Bar] | The width of the gap [mm] |
|--------------------------|---------------------------|
| 6                        | 0.083                     |
|                          | 0.069                     |
| 4                        | 0.054                     |
|                          | 0.043                     |
| 2                        | 0.028                     |
|                          | 0.022                     |

Figure 10 presents the exemplary views of measuring the width of the gap, located on top of a model, for 6 bars pressure.
4. Summary and analysis of research results

The performed studies were to verify the results obtained with strain gauge method and check the impact of multi-point pneumatic actuator on the connection glass-epoxy composite material with metal sheet using rivet nuts. Especially, focusing on the CAx simulations was important. An additional advantage of this article is to present the possibilities of modelling and simulation of endurance of composite materials in NX program. During the acquisition results of the simulations it was important to focus on the results of stress and displacement which were compared with the results of experimental tests. After the initial simulation space application forces and the place of mounting extensometer to experimentally measure were defined. The orientation of angles of the individual layers in the composite was shown, which had its influence throughout the test. After the simulation, and the results were entered into the tables and were compared with the results of experimental trials. Through simulation gap being formed on both sides of the uncured sample was visualized, which was difficult to measure during the strain gauge test. Analyising the obtained results displacements and stresses in both cases, i.e. computational method and experimental method specified percentage discrepancy between them. The difference in results for the metal sheet with holes was defined approximately at 9.1%, metal plate with rivets was defined to approximately at 6.7%, which is still the acceptable in adopted measurement error. The comparison of the results of the composite material was not as good as the percent difference which was calculated at 21%. It is a very large discrepancy that could be caused by a number of factors. One is the non-linearity of the form of the composite material. Although the simulation of the properties of the resin and the quantity was selected precisely according to the manufacturer's design intent, the sample was performed by hand. The stress of the difference between the computer and the experimental method was measured and was carried out on each extensometer, differences oscillate in the range of 1% to 31%. The difference in stress between the metal sheet and metal sheet with composite material equals 6% to 41%. Weight of the metal sheet during the test was 5 kg, and the weight of the composite sheet was 6 kg. The composite has increased mass by about 18% relatively to the total connection. However, the ratio of improve of the mechanical properties is greater than the weight and it can justify its use.

In conclusion the use of glass-epoxy composite has a positive impact on all mechanical properties as shown in the analysis of results. The composite can also serve as a protective layer for the connection of the metal sheet also increasing the resistance before the active and chemically aggressive substances. Gap created during the test involves the use of protective modes that will protect the entire connection from the inside. Used rivet nuts with the screw is a good releasable combination because it allows quick assembly and disassembly of worn parts of the composite and as
it was shown, it also has a positive effect on the mechanical properties. Towards its use also low price encourages.

Acknowledgments. The work was carried out under the project number PBS2/A6/17/2013 agreement implemented under the Applied Research Program, funded by the National Centre for Research and Development.

References
[1] Wróbel A 2012 Model of piezoelectric including material damping, Proceedings of 16th International Conference ModTech 2012, ISSN 2069-6736, pp 1061-1064.
[2] Wróbel A 2012 Kelvin Voigt’s model of single piezoelectric plate, Journal of Vibroengineering, Vol. 14, Issue 2, pp 534-537, ISSN 1392-8716.
[3] Buchacz A, Placzek M 2009 Damping of Mechanical Vibrations Using Piezoelements, Including Influence of Connection Layer's Properties on the Dynamic Characteristic, Solid State Phenomena, Volume: 147-149, pp 869-875.
[4] Białas K 2007 Synthesis of mechanical systems including passive or active elements reducing of vibrations. Journal of Achievements in Materials and Manufacturing Engineering, Volume 20, Issue 1-2, Gliwice, pp 323-326.
[5] Buchacz A, Placzek M 2012 The analysis of a composite beam with piezoelectric actuator based on the approximate method, Journal of Vibroengineering, Vol. 14, Issue 1, pp. 111-116, ISSN 1392-8716.
[6] Akul J, Ming L C 2006 NX5 for engineering design (Rolla: Department of Mechanical and Aerospace Engineering Missouri University of Science and Technology).
[7] Asundi A, Choi AYN 1997 Fiber metal laminates: an advanced material for future aircraft, Journal of Material Processing Technology 63, p. 384-394.
[8] Baier A, Majzner M 2011 Application of feature method to the modeling of composite structural elements. Journal of Achievements in Materials and Manufacturing Engineering, International OCOSCO World Press, 45, 1, p. 30-38.
[9] Dąbrowski H 2002 The strength polymer fiber composites (Wrocław: University of Technology Press).
[10] Jamroziak K, Bocian M 2006 Ballistic shields made of composite material on a polymer matrix. Technical magazine, Mechanics, Zeszyt nr 6-M/2006, Politechnika Krakowska, Kraków , s. 53-56.
[11] Świder J, Baier A, Majzner M 2010 Research and analysis of the properties of composites to the construction of wagons. Design and engineering structures, 11(38), s. 20-29.
[12] Buchacz A, Placzek M, Wróbel A 2013 Control of characteristics of mechatronic systems using piezoelectric materials, Journal of Theoretical and Applied Mechanics, Vol. 51 , pp. 225-234.
[13] Dymarek A, Dzitkowski T 2005 Modelling and synthesis of discrete – continuous subsystems of machines with damping. Journal of Materials Processing Technology, Vol. 164-165, p.1317-1326.
[14] Gwiazda A, Banaś W, Sękala A, Foit K, Hryniewicz P and Kost G 2015 Modular industrial robots as the tool of process automation in robotized manufacturing cells, 2015 IOP Conf. Ser.: Mater. Sci. Eng. 95 doi:10.1088/1757-899X/95/1/012104.
[15] Gwiazda A 2014 Adv. Mat. Res. 837 393-98 doi: 10.4028/www.scientific.net/AMR.837.393
[16] Rząsiński R. 2015 Technological similarity theory model for series of types of constructions, Special Issue: Evolution of modern manufacturing processes, technologies and materials, Int. J. Materials and Product Technology, Vol. 50, Nos. 3/4, pp 289 – 304.
[17] Oanta E, Panait C, Sabau A, Dumitrache C, Dascalescu A E 2015 Data filtration original algorithm for the computer based calculus of the stresses within an analytical model, International Journal of Modern Manufacturing Technologies, VII(2), 72-76.