Publishing House
AKAPIT

COMPUTER METHODS IN MATERIALS SCIENCE
Informatyka w Technologii Materiałów

Vol. 7, 2007, No. 1

VERIFICATION OF NUMERICAL MODELLING OF THE SPR JOINT BY EXPERIMENTAL STACK-UP

ROBERT CACKO, PIOTR CZYŻEWSKI

Institute of Materials Processing, Warsaw University of Technology
Narbutta 85, 02-524 Warszawa, Poland

Abstract

To analyze both joinability of various materials using self-piercing riveting (SPR) and strength of this kind of joints, full experimental and numerical procedure was introduced. It includes numerical investigation based on FEM analysis of riveting process, loading of already formed joint, experiments of SPR joining and the joint loading. The capability of proposed method by verification of selected cases is evaluated. Force progress during joint forming and joint loading in a standard shear test is the main comparison criteria between FEM simulations and experiments. Commercial software based on FEM: MSC.Marc and MSC.SuperForm are employed. Recently designed tool system and dies are used in experimental tests.

Key words: self-piercing riveting, SPR, FEM modelling, strength of the SPR joint

1. INTRODUCTION

In recent years self-piercing riveting (SPR) technique became one of the most promising method, especially in the automotive industry, for joining different materials. One of its main advantages was lack of several limitations traditionally associated with most widely used spot-welding. Self–piercing riveted joints also differ significantly from other types of riveted joints such as those used in aircraft structures owing to the manner in which the joint is produced. In self-piercing riveting, a tubular rivet made usually from a steel alloy is forced through a pair of partially overlapping sheets that are supported by a circular die with an axisymmetric cavity, figure 1. More details about this technique can be found e.g. in works by Neugebauer et al. (2000) and Barnes and Pashby (2000).

Although there are some successful industrial applications of the SPR, precise design criteria to obtain optimal strength of the joint are still needed.

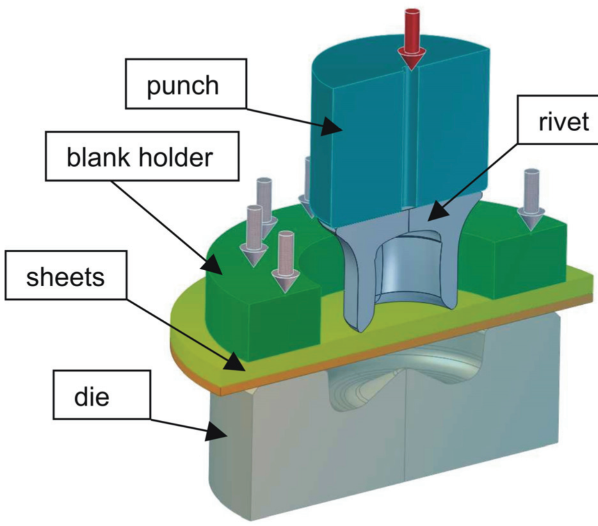

Fig. 1. Cross section of the SPR stack-up.

This fact limits this technology to be used more often. To have optimum strength of any SPR joint type (various materials, sheet thickness etc.), a set of analysis tools are needed. There are two components

 ISSN 1641-8581

of such an analysis system: widely verified numerical model, and experimental tooling system that allows quick joining of any shape. The industry expresses need for optimization of the SPR in two aspects: technological process optimization (symmetry of the joint assurance, production rate increase, etc.) and joint quality/properties improvement (strength of a joint increase/optimization, the bottom part of the joint shape and dimension control, decreasing the time of the process planning and die design for new materials and new designed stack-ups, etc). At the beginning of the SPR industrial application, so-called geometrical analysis was carried out based mainly on the industrial experiments. Nowadays, this approach can be supported by stress/strain analysis, which has stronger scientific background and can answer more general questions. Review of papers concerning the SPR method published recently point out, that all of them deal with either experimental tests of joining process, e.g. Wayne et al. (2005), as well as numerical and experimental tests of joining process, e.g. Porcaro et al. (2005) and Abea et al. (2006), or experimental tests of strength of a joint, e.g. Maofeng and Mallick (2003), Sun and Khaleel (2005) or Li and Fatemi (2006). Early works published by Cacko et al (2004) and Cacko and Czyżewski (2005a), also show visible progress in SPR modelling. In the Metal Forming Department, Warsaw University of Technology, a system for complex SPR analysis is introduced. It is based on three equivalent components:

- numerical analysis of forming a joint and loading it afterwards,
- laboratory set-up for the SPR joints forming with flexibility of changing dies,
- laboratory set-up for the SPR joint loading.

In current work a tryout of modelling and experimental verification of all aspects of the SPR technology concerning static procedure is presented. The first trial results were presented by Cacko and Czyżewski (2005b).

From among a set of various tests, two of them were chosen to demonstrate the results in different approaches. The first one started with tool design for a certain SPR case based on available general guidelines. Next, experimental tests of self-piercing riveting were carried out. After that, static loading tests were performed for strength estimation of formed joints. Finally, FEM analysis concerning joint forming and loading was done. The second test took opposite route. Numerical analysis was performed at the beginning to find out a joint without geometrical defects for a certain sheet thickness stack-up, and then experimental procedure was done to obtain geometry from FEM analysis.

2. EXPERIMENTAL TESTS

The special tool designed for the SPR is shown in figure 2. The main advantage of the tooling is that it can be easily assembled on almost any mechanical or hydraulic press. Special lower plate allows simple change of dies. Also dedicated dies were made for actual SPR cases. The profile of designed die for current tests is shown in figure 3. Standard rivet type 5x7 (5 mm tube part diameter and 7 mm total height) was used for validation tests. It is made of 10B35 (0.35% C) carbon steel, mechanically plated for high corrosion resistance, delivered as forged (hardness level around 320-340 HV). Coating is a combination of aluminium and zinc, which gives considerable corrosion resistance. Both upper and lower sheets are made of aluminium alloy 5052-H32 of 2mm thickness in first case, and 1mm each in the second one. Strength tests of joints were carried out on Instron machine equipped with precise force control. Examples of joints before and after shear tests are shown in figure 4.

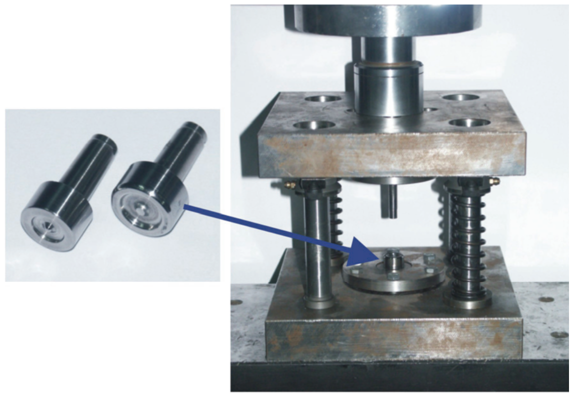

Fig. 2. *Dies and tool set-up for tests.*

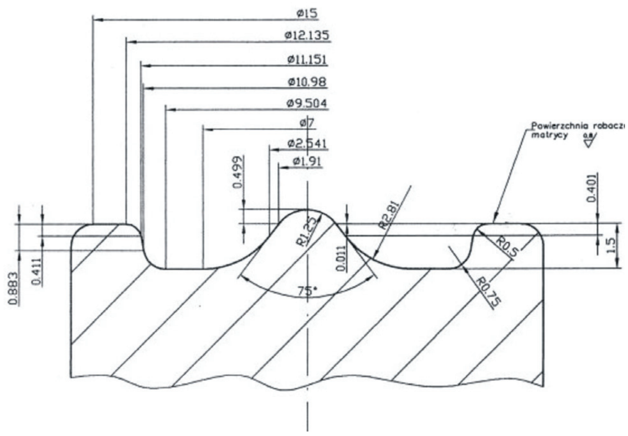

Fig. 3. *Die profile used in analysis.*

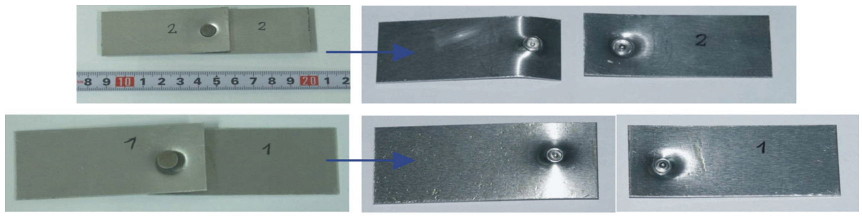

Fig. 4. Experimentally obtained SPR joints: left - after forming, right – after shear test.

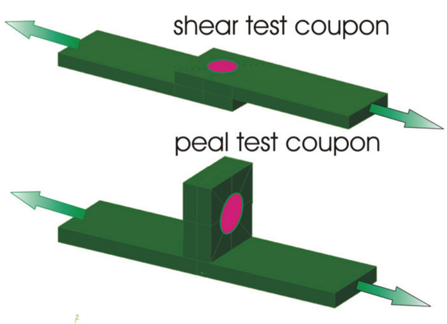

Fig. 5. Standard SPR joint loading tests.

3. FEM ANALYSIS

Both forming a joint and its strength prediction became the aims of FEM simulation, as the most promising tool, to analyse various aspects of the joint development. The mechanical response of a self-piercing riveted joint is determined by both the residual stresses after the piercing operation and the stress and displacement fields induced by the applied loads. To consider residual stresses due to piercing, two-dimensional analysis of joining process giving information on stress/strain field existing around a joint is performed. Then stress/strain field gained from 2D simulation is superimposed on 3D FEM model to analyze strength of a joint subjected to standard shear test shown in figure 5. Finally, experiments on originally designed tool stack – up are carried out. The force flow during joint formation and the force flow during loading a joint obtained by FEM simulations and experiments are compared. Numerical simulation of both forming SPR joint and modelling of loading/strength of a joint are performed by using commercial software MSC.Marc and MSC.SuperForm based on the FEM. Finally, comparison of the results of numerical simulations and experiments are presented.

As the problem of the SPR is axisymmetric, the four-node 2D axisymmetric elements have been used, with four Gauss points and a stiffened-based hourglass control (assumed strain co-rotational stiffness form). The size of the smallest element in both sheets and rivet was 0.1mm× 0.1mm. The punch and die were modelled as rigid bodies, while the material of the rivet and the sheets were modelled as elastic-plastic materials, assuming the von Mises yield criterion, a piecewise linear isotropic strain-hardening rule, and the associated flow rule in the plastic range. Table 1 collects material data for each deformable part of the model. Contact was modelled using an automatic 2D single-surface penalty formulation. Based on earlier publications, e.g. Cacko and Czyżewski (2004), friction coefficient was set on μ=0.3 for sheets and rivet interfaces. Other surfaces were modelled using μ=0.15.

Table 1. Material properties.

	Young modulus	Stress-strain relation	Yield stress
Rivet	210,000 MPa	$\sigma_p=639*\varepsilon^{0,246}$	σ_p^0=980MPa
Sheets	75,000 MPa	$\sigma_p=506,92*\varepsilon^{0,1899}$	σ_p^0=135MPa

The simulation runs in three steps. First, a displacement is prescribed for the blank holder. Then the punch pushes the rivet through the sheets until the joint is formed. Finally, a punch is released and springback analysis is performed in order to simulate the release of the tooling force. The initial and final stages of the riveting process are presented in figure 6.

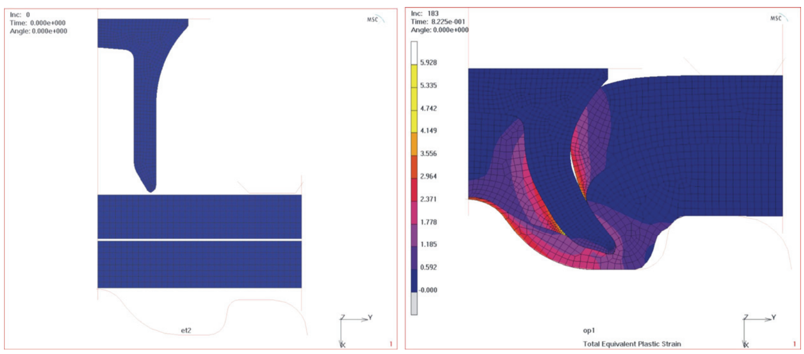

Fig. 6. Initial stage of SPR FEM simulation (left) and final joint shape (right).

Remeshing and element separation procedures are crucial for valid numerical simulation of the SPR joint forming. They were found to be considerably dependent on each other. Appropriate adjustment of their parameters is essential to have the problem of the upper layer piercing solved numerically. This is the software dependent factor since various analytical codes can use different approaches in both techniques. There are two separation procedures available in MSC software. Both are described briefly by Cacko and Czyżewski (2004). The procedure of tuning of remeshing and separation parameters for certain SPR cases in MSC.Marc and MSC.Superform were also described by Cacko and Czyżewski (2004). A procedure with element removal was applied in this paper.

stress/strain level, has to be made around created core. 8-node hexahedron elements are used in this case. The distance from the axis of symmetry that has to be taken into account during SPR simulation was prescribed by Cacko & Czyżewski (2005b). The rivet after loading is presented in figure 7b.

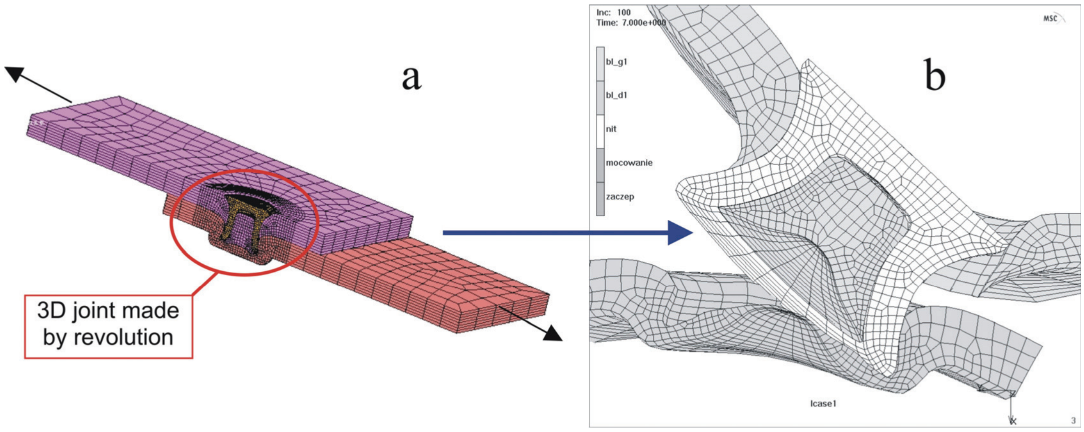

***Fig.** 7. 3D model for FEM simulation of loading (a) and stage of failure (b).*

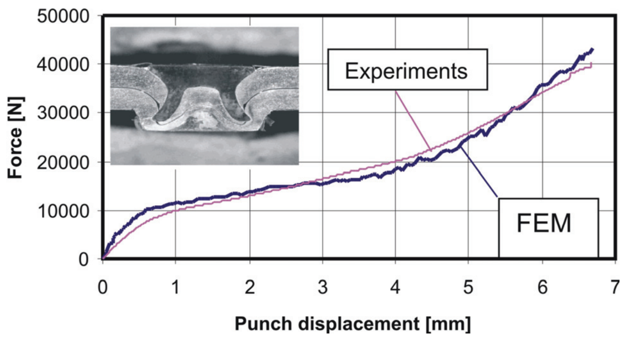

***Fig. 8.** Force comparison between FEM and experiments (2 mm sheets).*

Next stage of FEM analysis considered loading of a joint. That needed 3D model of a sample with a joint inside to be built, figure 7a. The procedure of stress/strain field transfer from 2D simulation over 3D model used in MSC.Marc was described by Cacko & Czyżewski (2005b). It proceeds in two stages. First, special „Axisymmetric model to 3D expand" procedure for expanding axisymmetric mesh to 3D is applied. Rotation angles and number of repetitions must be defined, where the latter means the number of elements created in circumferential direction. In this example, the 2D section was uniformly expanded over 180^0 in 12 sections. Then, the „Axisymmetric to 3D" route within „Initial conditions" module is used. This process converts stress/strain field results on 2D elements, to the new ones on the new 3D mesh. Once the 3D joint is developed, the rest of a sample, assumed to have zero

4. SUMMARY

Selected cases were chosen by a purpose. First, a stack-up with 2 mm sheets was examined. Then the same stack-up with 1 mm sheets was used, which was expected to give notably weaker joint. Comparison of the force progress between FEM analysis and experiments shows good accordance, figure 8. It is an important factor for precise analysis of joint formation process. Good agreement of these forces suggests, that conditions of joint forming are modelled correctly, so loading analysis should be valid. However, the comparison between FEM and experimental tests presented in figure 9 is more important for the SPR joint strength determination. The maximum force level was estimated below 5% difference for the 2mm sheets. The real variation in analyzed case between forces from experiments and numerical simulation appears after 4 mm displacement. The drop of the force observed in experiments can be explained by the misalignment of the rivet that was found after joint shape examination. The scheme of SPR failure – with long lasting plateau - prevents from rapid collapse anyway, so this divergence between FEM and experiments is not crucial for general joint strength estimation. Examination of the joint of 1mm sheets after loading showed some cracks of sheets' material. This kind of material failure was not taken into account in numerical model. It appears to be the reason why one may

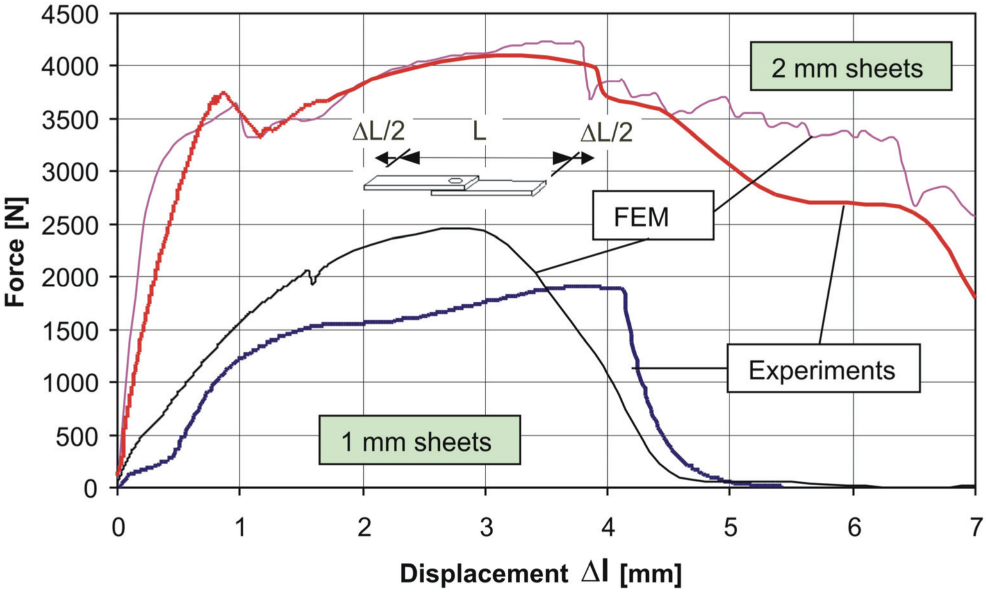

Fig. 9. *Comparison of loading force taken from FEM analysis and experiments (2mm sheets and 1mm sheets stack-ups).*

observe visible differences between force courses obtained by FEM and experiments. It could also explain why the joint from experiments is weaker than the one from simulation. Another important factor is the distance to complete failure, describing one of the advantages of riveted joints over spot-welded ones. Apparently this distance is modelled correctly according to experimental results, despite the differences in force prediction.

5. CONCLUSIONS

The procedure for complex SPR analysis, from joint forming up to strength prediction, is verified. The procedure works well for joints without cracks. Good agreement of the force obtained from FEM analysis and experiments is accomplished for riveting process and loading of 2mm sheets joint. Analysis of joining of 1 mm sheets with the same experimental stack-up shows that the numerical model needs to include updated failure criteria if prediction of force for not optimal joints with existing imperfections is needed. On the other hand, the second important factor – distance to failure – is modelled with a good accordance.

REFERENCES

Abea, Y., Kato, T., Mori, K., 2006, Joinability of aluminium alloy and mild steel sheets by self piercing rivet, *J. Mat. Proc. Techn.*, 177, 417–421.

Barnes, T.A., Pashby, I.R., 2000, Joining techniques for aluminium spaceframes used in automobiles. Part II – adhesive bonding and mechanical fasteners, *J. Mat. Proc. Techn.*, 99, 72-79.

Cacko, R., Czyżewski, P., Kocańda, A., 2004, Initial Optimization of Self-Piercing Riveting Process by Means of FEM, *Steel Grips, 2, Suppl. Metal Forming 2004*, 307-310.

Cacko R., Czyżewski P., 2005a, The method of self-piercing riveting (SPR) modelling, *Proc. KomPlasTech 2005 Conf.*, Ustroń, eds, Piela, A., Lisok, J., Grosman, F., 73-81 (in Polish).

Cacko R., Czyżewski P., 2005b, Numerical modelling of forming and loading SPR joints, *Przegląd Mechaniczny*, 7-8, 50-53 (in Polish).

Li, B., Fatemi, A., 2006, An experimental investigation of deformation and fatigue behaviour of coach peel riveted joints, *Int. J. of Fatigue*, 28, 9–18.

Maofeng, Fu, Mallick, P.K., 2003, Fatigue of self-piercing riveted joints in aluminium alloy 6111, *Int. J. of Fatigue*, 25, 183–189.

Neugebauer, R., Mauermann, R., Dietrich, S., 2000, Spot-Joining – New Technologies in the Field of Joining by Forming, *Advanced Technology of Plasticity*, Vol. 2, 41-51.

Porcaro, R., Hanssen, A.G., Langseth, M., Aalberg A., 2006, Self-piercing riveting process: An experimental and numerical investigation, *J. Mat. Proc. Techn.*, 171, 10–20.

Sun, X., Khaleel, M.A., 2005, Strength estimation of self-piercing rivets using lower bound limit load analysis, *Science and Technology of Welding and Joining*, 10, 624- 535.

Wayne Cai, P.C. Wang, Wu Yang, 2005, Assembly dimensional prediction for self-piercing riveted aluminium panels, *Int. J. of Machine Tools & Manufacture*, 45, 695– 704.

NUMERYCZNE I EKSPERYMENTALNE BADANIE TWORZENIA I OBCIĄŻANIA POŁĄCZEŃ NITOWANYCH BEZOTWOROWO - SPR

Streszczenie

W ostatnich latach nitowanie bezotworowe – self-piercing riveting (SPR) – staje się bardzo obiecującą techniką, zwłaszcza w przemyśle motoryzacyjnym, do łączenia blach z różnych materiałów. Jego atrakcyjność wynika głównie z powodu braku ograniczeń tradycyjnie związanych z najczęściej stosowanym do łączenia elementów cienkich zgrzewaniem punktowym. Przy zachowaniu cech tradycyjnego nitowania metoda SPR wyróżnia się sposobem powstawania połączenia. W nitowaniu bezotworowym nit o specjalnym kształcie, zwykle ze stali lub stopów aluminium, jest wciskany w dwie – lub więcej – częściowo zachodzące na siebie warstwy. Do uzyskania połączenia jest ponadto wymagana matryca o specjalnie zaprojektowanym profilu, w który wpływa materiał łączonych blach i odkształcanego nitu. Odpowiednie dopasowanie kształtu matrycy do grubości blach, kształtu nitu oraz rodzaju materiałów stanowi największy problem w optymalizacji jakości połączenia. Chociaż metoda SPR jest już stosowana w przemyśle z częściowym sukcesem, wciąż brak jest jednoznacznych kryteriów projektowych, które umożliwiałyby w miarę szybkie optymalne projektowanie połączeń oraz pełniejsze wykorzystanie potencjału metody. W Zakładzie Obróbki Plastycznej Instytutu Technologii Materiałowych Politechniki Warszawskiej określono dwa kierunki badań metody SPR mające na celu zastosowanie modelowania numerycznego do określenia głównych czynników wpływających na optymalną wytrzymałość połączenia, oraz zaproponowanie modyfikacji nitowania bezotworowego do tworzenia mikropołączeń. Możliwości modelowania kształtowania połączenia jak i oceny jego wytrzymałości na drodze symulacji komputerowych zostały przez autorów wstępnie przetestowane. Obecnie poddano weryfikacji oryginalnie zaprojektowane stanowisko badawcze do wykonywania połączeń SPR.

Wytrzymałość połączenia SPR jest związana z polem naprężeń pozostałym w strefie wokół połączenia po procesie kształtowania. W poniższej pracy przestawiono wybrane wyniki trójwymiarowego modelowania obciążania połączeń i porównania wyników symulacji z wynikami badań laboratoryjnych dla wybranych zestawów. W celu uwzględnienia stanu naprężeń w modelu MES powstałego po procesie nitowania przeprowadzono symulacje numeryczne osiowosymetrycznego modelu 2D. Następnie stan odkształceń/naprężeń został przeniesiony za pomocą specjalnej procedury do modelu trójwymiarowego. W dalszej kolejności poddano obciążaniu połączenia według jednego ze standardowych testów. Porównywano kształt połączeń oraz przebiegi sił podczas nitowania i obciążania połączeń uzyskane na drodze symulacji i z testów laboratoryjnych. Do obliczeń numerycznych wykorzystano komercyjne oprogramowanie firmy MSC – MSC.Marc i MSC.SuperForm – oparte na metodzie elementów skończonych.

Submitted: September 28, 2006
Submitted in a revised form: October 26, 2006
Accepted: October 30, 2006