УДК 621.981.02

Naser M. Elkhmri, Budar Mohamed R.F.
College of Engineering Technology – Janzour, Tripoli, Libya
Hamza Abobakr O.
Technical College of Civil Aviation & Meteorology, Tripoli, Libya
Igor Shepelenko, PhD
Central Ukrainian National Technical University, Kropyvnytskyi, Ukraine
E-mail: *kntucpfzk@gmail.com*

Comparison between Experimental and Simulation Results of Bending Extruded Aluminum Profile

This paper focuses on the reliability of vehicle body structure made of aluminum alloy, it also investigates the influence of bending process on strength and forming quality of the part. The optimization of profile shape section during bending process is very crucial factor for the component reliability and forming quality. However, it is difficult to get ideal solution for optimization of forming process by using only theoretical analysis and experiments. Therefore numerical simulation based on finite element is used more and more to analyze the forming mechanism, to predict the defects and to optimize the process parameters. ABAQUS software shows detailed preprocessing steps of the profile design and formation followed by an accurate model to work on. The simulations allow investigating critical areas of the profile during the formation process. This paper studied the evolution of thickness deformation along the profile, the middle section was targeted since it is the most deformed area, then compared the variations of thickness between simulation and experiments in all cases using punch only and die-punch. The agreement between the experiments and the simulations is fairly good for most cases, which proves the validity of the established numerical model.
aluminum profile, bending process, numerical simulation, ABAQUS software

Насер М. Елкхмри, Будар Мохамед Р.Ф.
Инженерно-технологический колледж, Триполи, Ливия
Хамза Абобакр О.
Технический колледж гражданской авиации и метеорологии, Триполи, Ливия
И.В.Шепеленко, доц., канд. техн. наук
Центральноукраинский национальный технический университет, Кропивницкий, Украина

Сравнение результатов экспериментов и расчетов моделирования изгиба штампованного алюминиевого профиля

В статье исследованы надежность конструкции кузова автомобиля из алюминиевого сплава, а также влияние процесса изгиба на прочность и качество штамповки детали. Оптимизация формы сечения профиля в процессе изгиба является важным фактором повышения надежности и качества изготовления деталей. Однако получение идеального решения при оптимизации процесса изгиба представляется сложным, используя лишь теоретический анализ и результаты экспериментов. Для анализа механизма формирования, прогнозирования дефектов и оптимизации параметров процесса изгиба возможно применение численного моделирования на основе метода конечных элементов. Программное обеспечение ABAQUS показывает этапы проектирования предварительной обработки профиля с последующим получением рабочей модели. Моделирование позволяет исследовать критические участки профиля в процессе их формирования. В статье исследовались наиболее деформированные области средней части профиля путем сравнения результатов моделирования и экспериментов. Корреляция между результатами моделирования и экспериментов доказывает обоснованность установленной численной модели.
алюминиевый профиль, процесс изгиба, численное моделирование, программное обеспечение ABAQUS

© Naser M. Elkhmri, Budar Mohamed R.F., Hamza Abobakr O., Igor Shepelenko, 2017

Introduction. In recent years aluminum profile is widely used in various fields such as aeronautics and automobile industries to get light product. As one of main forming process to obtain the profile part, bending becomes the major development direction of advanced plastic forming technology. The materials science, mechanics and computer application for manufacturing technology peristaltic the process of plastic forming technology and advanced numerical simulation [1]. The development of advanced forming technology in the promotion of science and technology, economic and defense modernization has an important role in the process. In the process of plastic bending profile, the product generally desire to achieve lightweight, high strength, high precision, high efficiency, low consumption and other requirements [2]. For this 21st century the plastic processing technology [3], research and development has advanced enormously, most developed for the automotive, aviation, and aerospace industry and has bringing a broad application prospects [4].

Source of project. In factory, the manufacturing process of parts such as machining, forming and welding will affect on the quality and service life of parts. Forming local of plate parts such as excessive thinning caused defect thickness deformation and cracks. Affecting the weld defects and temperature on the welding material properties and other factors such as local forming and welding deformation, will not only affect the appearance, will change the stress state and causing the error theory calculation model. So, the equation is; how to reduce or avoid these defects? To study this forming bending process is a meaningful subject.

This paper is based on above background, thickness deformation of bending extruded aluminum alloy profile, molded parts and analysis of the causes of defects is aim project research, the forming process of form and position at the same time of aluminum alloy profile into pieces to study thickness deformation on the part during the operation, explore and elimination of defect provides the reference for the design and manufacture of molding parts.

The object of study. The goal of this study is to optimize the bending process parameters and the profile section shape using numerical simulation. The simulation is realized using the finite element method software ABAQUS. The main contents of research work include ABAQUS, which uses to obtain an accurate model to work. The simulation gives information about the location of damage areas on the profile after bending. Also studied the evolution of thickness deformation along the profile, the middle section of profile was targeted since it is the area the most deformed. And compared the variations of thickness deformation between simulation and experiments results. There are similarities, but the values from the experiments are not accurate enough, and the simulation suffers from edge effect.

Analysis of finite element simulation. Generally, finite element simulation process is divided into the following;

Explicit algorithm. The explicit calculation also known as the central difference method, acceleration and velocity has the following hypothesis;

$$\ddot{u}_t = \frac{1}{\Delta t^2}\left(u_{t-\Delta t} - 2u_t + u_{t+\Delta t}\right) \quad (1)$$

$$\dot{u}_t = \frac{1}{2\Delta t}\left(u_{t-\Delta t} - u_{t-\Delta t}\right) \quad (2)$$

The two above-described relations is substituted into the equation of motion to get central difference method of recursive formula as follows;

$$\left(\frac{1}{\Delta t^2}M + \frac{1}{\Delta t}C\right)u_{t+\Delta t} = Q_t - \left(K_t - \frac{2}{\Delta t^2}M\right)u_t - \left(\frac{1}{\Delta t^2}M - \frac{1}{2\Delta t}C\right)u_{t-\Delta t} \quad (3)$$

From above equation, at the current time only the displacement acceleration is related to the displacement of the previous time, the only means of solving current moment without iterative process [5].

Implicit algorithm. The implicit algorithm is relatively complicated, the widely used method is Newmark, the relationship between acceleration, velocity and displacement as follows;

$$\dot{u}_{t+\Delta t} = \dot{u}_t + \left[(1-\delta)\ddot{u}_t + \delta\, \ddot{u}_{t+\Delta t} \right] \Delta t \ , \quad (4)$$

$$u_{t+\Delta t} = u_t + \dot{u}_t\, \Delta t + \left[\left(\frac{1}{2} - \alpha \right) \ddot{u}_t + \alpha\, \ddot{u}_{t+\Delta t} \right] \Delta t^2 \ . \quad (5)$$

Substituted into the equations of motion can be obtained recursive formula as following;

$$\left(K_{t+\Delta t} + \frac{1}{\alpha \Delta t^2} M + \frac{\delta}{\alpha \Delta t} C \right) u_{t+\Delta t} = Q_{t+\Delta t} + M \left[\frac{1}{\alpha \Delta t^2} u_t + \frac{1}{\alpha \Delta t} \dot{u} + \left(\frac{1}{2\alpha} - 1 \right) \ddot{u}_t \right]$$
$$+ C \left[\frac{\delta}{\alpha \Delta t} u_t + \left(\frac{\delta}{\alpha} - 1 \right) \dot{u} + \Delta t \left(\frac{\delta}{2\alpha} - 1 \right) \ddot{u}_t \right] \ . \quad (6)$$

From previous equation, the displacement, velocity and acceleration at any time are related to each other, this makes solving the equations of motion into a series of solving nonlinear equations, must be achieved through an iterative and simultaneous equations [6]. There are two problems need to be solved in the process of solution: first, is not necessarily can convergence of iterative process; second, simultaneous equations is possible to determine solution without terrible appearance. Therefore, for solving nonlinear implicit algorithm cannot guarantee the convergence. In addition, at each step of the calculation require iterative solution for equation of static equilibrium, the need to occupy a considerable amount of computing resources, disk space and memory [7].

ABAQUS software. ABAQUS/CAE is a finite element software, used complex system for modeling constraints and deformations [8]. It is composed of different modules; every module helps to describe the system. It is widely used in the vehicle, aerospace and industrial products. The product is popular with academic and research institutions due to the wide material modeling capability and the program's ability to be customized [9]. ABAQUS also provides a good collection of Multiphysics capabilities, such as coupled acoustic structural, piezoelectric and structural pore capabilities, making it attractive for production level simulations where needs multiple fields to be coupled.

Each complete finite element analysis depend on 3 separate stages;

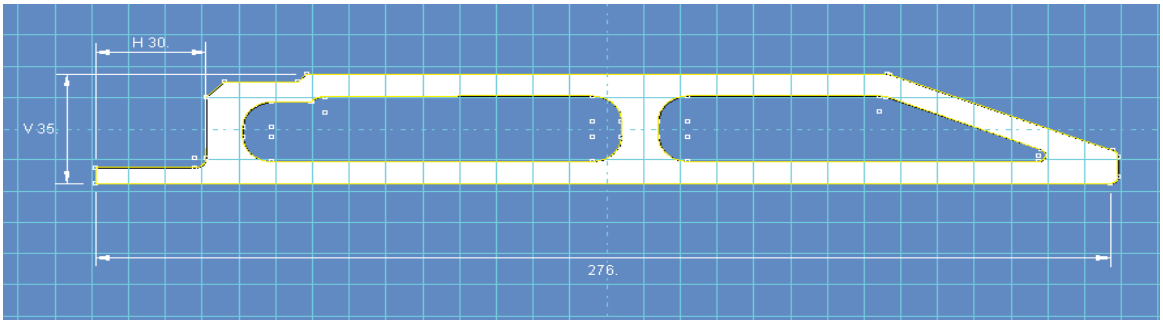
I. Pre-processing or modeling: This stage involves creating an input file which contains an engineer's design for a finite element analyzer.

II. Processing or finite element analysis: This stage produces an output visual file.

III. Post-processing or generating report, image, animation, etc. from the output file: This stage is a visual rendering stage.

Bending forming simulation & experimental verification

Characteristic of the profile part. In this paper we will work on vehicle structure profile as shown in the Picture 1, which made of aluminum alloy A6N01S with mechanical properties as shown in the table 1.



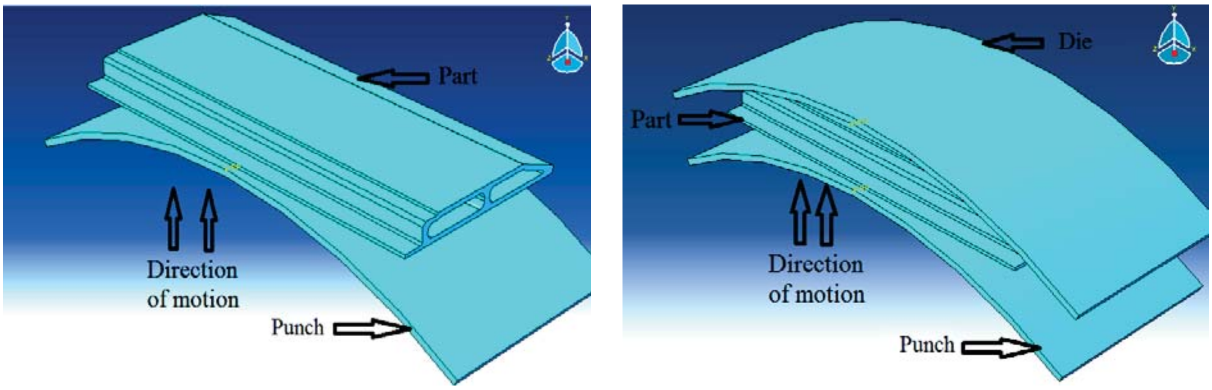
Picture 1 – Profile section size

Table 1 - Mechanical properties

Modulus of elasticity E / MPa	Yield strength $\sigma_{0.2}$/MPa	Poisson's ratio	Tensile strength (Mpa)	Strength coefficient K (Mpa)	Strain hardening Index n	Elongation %
66946	232.211	0.359	265.843	343.910	0.079 8	8

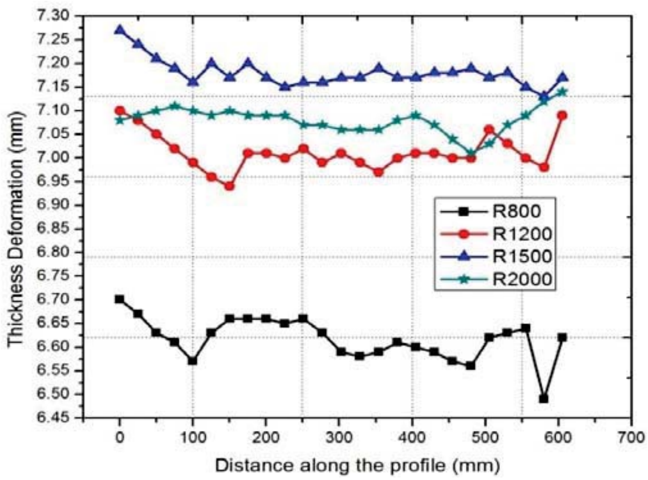
Many factors that affect the actual formation process, is not likely to be quite successful consistent with the application of the program, it is very complex. From here, to describe the process faster and more accurately, we choose the operation parameters is the key to control.

In this paper, the attention is paid to forming process and influence of die radius, therefore, in order to improve the computational accuracy and efficiency goals will use punch only & die-punch as shown in the Picture 2, by Using ABAQUS/CAE software, apply simulation process on the part such as (properties, assembly, step, interaction, load and mesh) can find the optimization amount of thickness deformation of the profile due to bending process for four types of radiuses (R800 – R1200 – R1500 – R2000) mm.



Picture 2 – Methods of bending

In the Picture 3, shown the thickness deformation along the profile of experimental results for all radiuses (R800 - R1200 - R1500 - R2000), where we can see from the graph most deformation is done at the ends of the profile, and the maximum thickness deformation for each radius is shown in the table 2, which notes that whenever die's radius increases, the value of thickness deformation decreases.

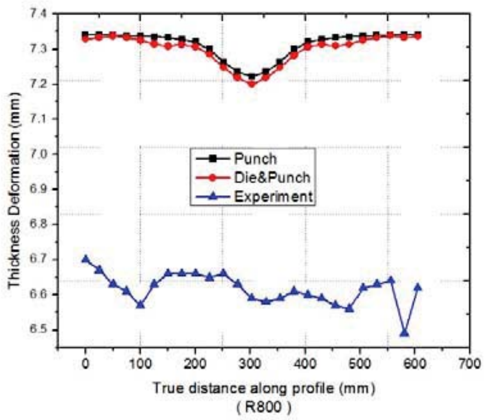


Picture 3 – Thickness deformation of experimental results for all radiuses

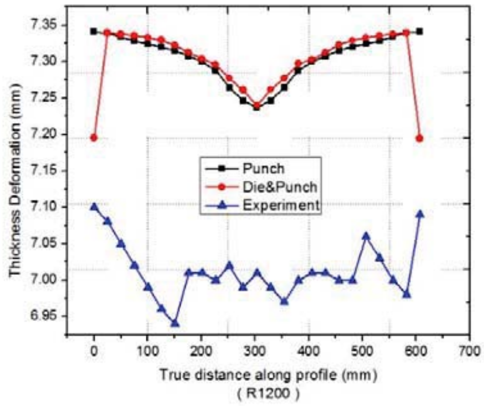
Table 2 - Experimental thickness deformation

Radius R (mm)	800	1200	1500	2000
Thickness Deformation (mm)	0.21	0.16	0.14	0.13

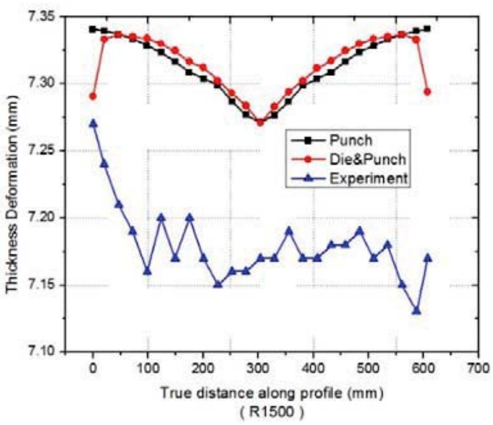
Comparison between thickness deformation of experimental results with thickness deformation of simulation results for each radius in punch only & die-punch.



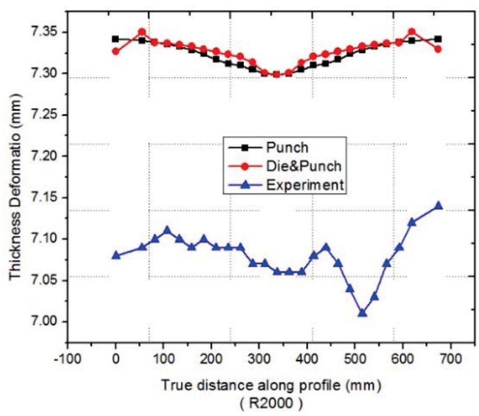
Picture 4 – Thickness deformation for radius (R800)



Picture 5 – Thickness deformation for radius (R1200)



Picture 6 – Thickness deformation for radius (R1500)

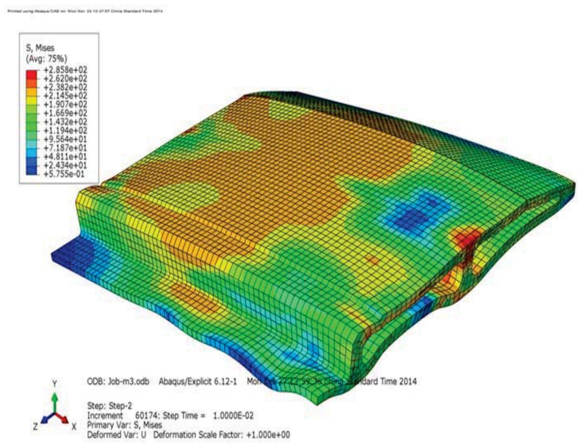


Picture 7 – Thickness deformation for radius (R2000)

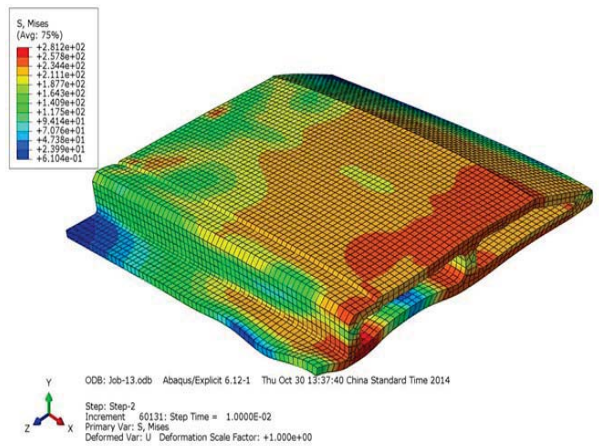
By focusing on the previous graphs, in case of radius R800 most thickness deformation done in the middle of profile. In bending process of radius R1200 and R1500, max. thickness deformation is occur in the middle of the profile when using punch only, while this deformation occur in the middle and ends of the profile in case of using die-punch. In radius R2000, when using punch only a small amount of thickness deformation done in middle of the profile, but in case of using die-punch the thickness deformation done at the ends and middle of profile, this refers to the contact between the punch and profile during bending process. In all cases, we note the amount of thickness deformation decreases whenever bending radius increase whether using punch only or using die-punch.

Table 3 - Max. Thickness Deformation of Simulation Results

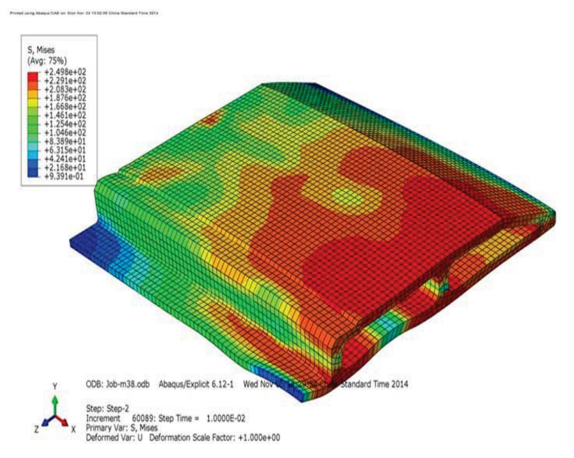
	R800		R1200		R1500		R2000	
Max. Thickness Deformation (mm)	Punch	Die-Punch	Punch	Die-Punch	Punch	Die-Punch	Punch	Die-Punch
	0.1204	0.1390	0.1045	0.1450	0.0697	0.0664	0.0430	0.0519



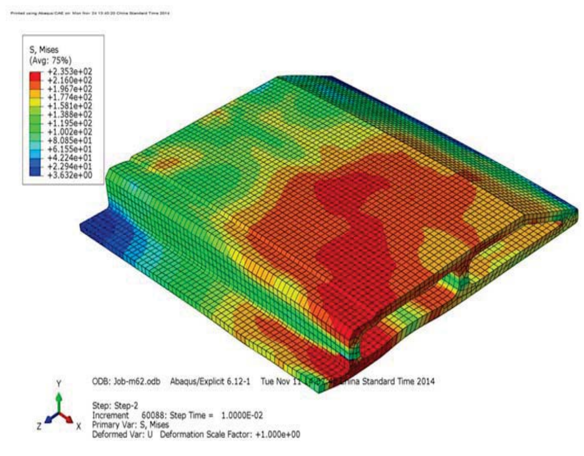
Picture 8 – Cross section in the middle of profile (R800)



Picture 9 - Cross section in the middle of profile (R1200)



Picture 10 - Cross section in the middle of profile (R1500)



Picture 11 - Cross section in the middle of profile (R2000)

When cut the profile from the middle, we notices clearly a distortion in the middle area during bending process as shown in the figures 8, 9, 10 and 11. The amount of distortion is greatest in case of R800 and this amount becomes less when bending radius becomes bigger. Also this distortion causes irregularity in the external shape of the profile, this may cause problems during final assembly.

Conclusion. In this paper, permits to visualize the bending process of extruded aluminum profile, to know the problems that occur after bending process of the profile, such as thickness deformation and also, when cutting the profile from the middle, notes the presence of distortions in the internal shape of the profile due to bending process, this distortion increases whenever the radius decreases. Through the comparison between the experiments and simulation results, we can conclude that the optimization design which includes all the suitable parameters and in line with the actual manufacturing process of the profile, where faces less problems to obtain the best values of thickness deformation, when using die-punch of radius R1200.

Recommendation. Recommend to use ABAQUS software, its provide very convenient ways for modeling the profile. The simulation had to give us information about the location of critical area on the profile after bending process. It also helps to save money, time and effort to obtain the optimization results, which could be applied to the actual product in the industry.

To avoid distortion in the middle of the profile, we recommended to change the inner design of the profile with add support columns which help to reduce the amount of deformation taking into account that the weight and outer design of the profile are not changed.

References

1. Wang Yan Jin, DING Guo-hua, Wang Jun Nine, Aluminum car body manufacturing technology development status and prospects in China [J], welding, 2004 (10): 5-7.
2. Wu Liang group, sinus new, CSR Sifang: Road of independent innovation of high-speed train [J], to the world, 2008, 07: 50-54.
3. Mykhailo Chernovol. Appliances for FANT of cylinder liners/ Mykhailo Chernovol, Igor Shepelenko, Budar Mohamed R.F. // Збірник наукових праць Кіровоградського національного технічного університету «Конструювання, виробництво та експлуатація сільськогосподарських машин». 2016. – Вип.46. – С. 258-265.
4. Lvbi Feng, China's new generation of high-speed motor applications and integration manufacturing process technology [J], technological innovation and Herald, 2010, No.16: 85-87.
5. Miao Jin. Aluminum extrusions, sheet stretch forming simulation and contact problems. Beijing University of Aeronautics Postdoctoral Research Report, 2003.
6. Opheim B S. Bending of Thin-Walled Aluminum Extrusions [D]. Trondheim, Norway: Department of Structural Engineering, the Norwegian University of Science and Technology, 1996.
7. Lademo O G., Hopperstad O S, Pedersen K O. Modeling of plastic anisotropy in heat-treated aluminum extrusions[J], Journal of Material Processing Technology, 2002, 125-126.
8. Tryland T., Hopperstad O S, Langseth M. Design of experiments to identify material properties [J]. Materials and Design, 2000, 21:477-492.
9. Li S, Reid S R. Relationship between the elastic buckling of square tubes and rectangular plates [J]. Journal of Applied Mechanics, 1990, 57: 969.

Насер М. Елкхмрі, Будар Мохамед Р.Ф.
Інженерно-технологічний коледж, Тріполі, Лівія
Хамза Абобакр О.
Технічний коледж цивільної авіації та метеорології, Тріполі, Лівія
І.В. Шепеленко, доц., канд. техн. наук
Центральноукраїнський національний технічний університет, Кропивницький, Україна

Порівняння результатів експериментів і розрахунків моделювання вигину штампованого алюмінієвого профілю

У статті досліджені надійність конструкції кузова автомобіля з алюмінієвого сплаву, а також вплив процесу вигину на міцність та якість штампування деталі. Оптимізація форми перетину профілю в процесі вигину є важливим чинником підвищення надійності та якості виготовлення деталей. Метою даної роботи є оптимізація параметрів процесу вигину штампованого алюмінієвого профілю, яка реалізовувалася чисельним моделюванням з використанням програмного забезпечення ABAQUS.

Програмне забезпечення ABAQUS засноване на методі кінцевих елементів для виконання віртуальних випробувань з використанням реалістичного моделювання, що допомагає скоротити витрати й час на розробку продукту, а також підвищити надійність виробів, що випускаються у машинобудуванні. Моделювання дозволяє знаходити та досліджувати критичні ділянки профілю в процесі їх формування. Використання програмного комплексу ABAQUS дозволило здійснити візуалізацію процесу вигину алюмінієвого профілю, дослідити дефекти, що виникають у процесі вигину профілю. Розглянуто найбільш деформовані області середньої частини профілю шляхом порівняння результатів моделювання й експериментів. Кореляція між результатами моделювання й експериментів доводить обґрунтованість встановленої чисельної моделі.

Для підвищення міцності алюмінієвого профілю рекомендовано змінити його внутрішню конструкцію, при цьому вага й дизайн профілю залишилися незмінним.
алюмінієвий профіль, процес вигину, чисельне моделювання, програмне забезпечення ABAQUS

Одержано 11.05.17

УДК 539.3

А.Ю. Лысых, канд. техн. наук
Первомайский политехнический институт НУК им. адмирала Макарова, г.Первомайск, Украина
E-mail: kobmargarita@mail.ru

Пространственное деформирование гибких конструкций, соединяющих судно-носитель и присоединенный объект

Приводится метод численного решения задачи колебания гибкого соединяющего звена морского назначения, основанный на совместном применении метода продолжения по параметру и метода Ньютона-Канторовича. Описаны способы приложения динамических нагрузок и постановки граничных условий. Приведены результаты решения задачи.
упругое деформирование, гибкий элемент, численные методы

А.Ю. Лисих, канд. техн. наук
Первомайський політехнічний інститут НУК ім. адмірала Макарова, м.Первомайськ, Україна
Просторова деформація гнучких конструкцій, які з'єднують судно-носій и приєднаний об'єкт
Розглянуто метод числового розв'язання задачі коливання гнучкої сполучаючої ланки морського призначення, побудований на сумісному застосуванні методу продовження по параметру і методу Ньютона-Канторовіча. Описані способи додавання динамічних навантажень і постановки граничних умов. Наведені результати рішення задачі.
пружна деформація, гнучкий елемент, чисельні методи

Постановка проблемы. Проблема транспортировки нефте- и газопродуктов с судна-носителя (СН) к присоединенному объекту (ПО) стала в последнее время особенно актуальной. Это обуславливает необходимость разработки и создания новых типов гибких конструкций, повышения их прочности и надежности. Таким соединяющим звеном могут быть гибкие армированные шланги, гибкие трубопроводы постоянной жесткости, гибкие бандажированные трубопроводы и т. п.

© А.Ю. Лысых, 2017