JOURNAL OF MARINE SCIENCES & ENVIRONMENTAL TECHNOLOGIES
Vol. 8, Issue No. 2 (December-2022)
ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

FEM Simulation as a Tool for Planning and Optimizing the Rubber Pad Forming Process

Muamar M. Benisa[1, *], Galal H. Senussi[2], Abdalla S. Tawengi[3], and Abdelnasir M. Shtewi[4]

[1] Mechanical Engineering Department, Faculty of Engineering, Alasmarya Islamic University, Zliten, Libya.
[2] Mechanical Engineering Department, Omar Al-Mukhtar University, Elabayda, Libya.
[3] Faculty of Oil and Gas Engineering, Zawia University, Zawia, Libya.
[4] Libyan authority for Scientific Research, Tripoli, Libya.

[*] **Corresponding author:** m.benisa@asmarya.edu.ly

محاكاة العناصر المحدودة كأداة لتخطيط وتحسين عملية التشكيل بالوسادة المطاطية

معمر بن عيسى[1، *]، جلال السنوسي[2]، عبدالله الطاونجي[3]، عبدالناصر الشتيوي[4]

[1] قسم الهندسة الميكانيكية، كلية الهندسة، الجامعة الأسمرية الإسلامية، زليتن، ليبيا.
[2] قسم الهندسة الكيميائية، كلية الهندسة، جامعة عمر المختار، البيضاء، ليبيا.
[3] كلية النفط والغاز، جامعة الزاوية، الزاوية، ليبيا.
[4] الهيئة الليبية للبحث العلمي، طرابلس، ليبيا.

Received: 10 November 2022; Revised: 15 December 2022; Accepted: 27 December 2022.

Abstract

Manufacturing press-formed metallic components by conventional methods needs a burdensome trial-and-error process to set up the technology, which success depends, largely, upon the operator's skill and experience. The finite element (FE) simulations of sheet-metal-forming processes assist the manufacturing engineer to design a forming process by shifting the costly press-shop try-outs to the computer-aided design environment. The purpose of applying numerical simulations of a manufacturing process such as rubber-pad forming is to avoid the trial-and-error procedure and shorten the development phases when tight times-to-market are demanded. The main aim of the investigation presented in this paper was to develop a numerical model that would be able to, successfully, simulate a rubber-pad forming process. The finite-element method was used for blank- and rubber-behavior predictions during the process. The study focused on simulating and investigating significant parameters (such as forming force and stress and strain distribution in a blank) which are associated with the rubber-pad forming process, also the capabilities of this process regarding the manufacturing of aircraft wing ribs and ships. As a result, the stress and strain distribution in a blank as well as the forming force were identified. The experimental analysis of a rib with a lightning hole showed a close correlation between the FE simulations and the experimental results.

Keywords: *FE simulation; Optimization; Rubber pad forming; Sheet metal forming,*

الملخص

يحتاج تصنيع المكونات المعدنية المشكلة بالضغط بالطرق التقليدية إلى عملية تجربة وخطأ المرهقة من أجل إعداد التكنولوجيا التشكيل، حيث يعتمد النجاح، إلى حد كبير، على مهارة المشغل وخبرته. تساعد محاكاة العناصر المحدودة (FE) لعمليات تشكيل الصفائح المعدنية مهندس التصنيع في تصميم عملية التشكيل عن طريق تحويل تجارب الضغط المكلفة إلى بيئة التصميم بمساعدة الكمبيوتر. الغرض من تطبيق المحاكاة العددية لعملية التصنيع مثل

تشكيل الوسادة المطاطية هو تجنب إجراء التجربة والخطأ وتقصير مراحل التطوير عند ضيق أوقات الطلب في السوق. الهدف الرئيسي من الدراسة الحالية هو تطوير نموذج رقمي يمكنه محاكاة عملية تشكيل بالوسادة المطاطية بنجاح. تم استخدام طريقة العناصر المحدودة لتنبؤ بسلوك المعدن (الصفيحة) والوسادة المطاطية أثناء العملية. ركزت الدراسة على محاكاة واستقصاء العوامل المهمة (مثل: القوة، التشكيل، وتوزيع الإجهادات والانفعالات على المعدن) المرتبطة بعملية تشكيل الوسادة المطاطية، وكذلك إمكانية هذه العملية فيما يتعلق بتصنيع أضلاع (عروق)جناح الطائرة والسفن. كنتيجة لذلك، تم تحديد توزيع الإجهاد والانفعال في المعدن بالإضافة إلى قوة التشكيل. وبينت التحاليل التجريبية للضلع (عرق) بالفتحة وجود علاقة وثيقة بين محاكاة FE والنتائج التجريبية.

الكلمات الدالة: محاكاة العناصر المحدودة؛ التحسين؛ تشكيل الوسادة المطاطية؛ تشكيل الصفائح.

1. Introduction

Stamping is a metal forming process, in which the sheet metal is punched using a pressing tool that is loaded on a machine or a stamping press in order to form the sheet into the desired final shape (Fig. 1). The conventional stamping process is performed through a punch and a blank holder, which works on forcing the sheet metal to slide into a die and comply with its shape. Rubber-pad forming is a metalwork process in which sheet metal is pressed between a die and a rubber block. Generally, an elastic upper die, usually made of rubber, would be connected to a hydraulic press. While a rigid lower die, often called a form block, provides the mold used in forming the sheet metal. Due to the fact that the upper die (male) can be used with separate lower dies (female), the process is relatively cheap and flexible. However, under the same circumstances, rubber pads exert less pressure than non-elastic parts, which might lead to less definition in forming or to defects appearance. Using the rubber-pad forming process instead of the conventional metallic tools has a number of advantages, such as: i) the same flexible pad could be used to form several different work piece shapes, and that comes as a result, the rubber pad has the ability to return to its original shape after each process; ii) the tool costs are lower compared to the conventional forming processes; iii) the thinning of the work metal, which occurs in the conventional deep drawing is reduced considerably; iv) the set-up time can be, considerably, reduced in this kind of processes, where there are no die clearance or alignment checks need to be made; and v) although no lubrication is needed, the good surface finish could be achieved, where no tool marks are created.

However, the rubber-pad forming processes have several disadvantages, such as: i) the lifetime of the flexible pad will be limited (depending on the severity of forming combined with the pressure level); ii) the lack of a sufficient forming pressure results in parts with less sharpness or with wrinkle, which leads to reworking the part to its correct shape and dimensions; iii) low production rate, so this process is suitable, mostly, for small series (typical of the aircraft industry); and iv) the dependency of the final design on the exact geometry of the tool (ASM, 2006; Dirikolu & Akdemir, 2004; Sala, 2001; Takuda & Hatta, 1998; and Thiruvarudchelvan, 1993).

For these reasons, it is recommended to use the finite element simulation of the manufacturing process during the conceptual design. Moreover, it could give important answers in analyzing the process and predicting the defects that may occur. Therefore, modifications could be

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

easily performed prior to tool manufacturing and part production. This study presents a numerical simulation of a rubber pad forming process of an aluminum alloy with different tool geometry for supporting ribs of an aircraft tail, optimized in order to find out the best design of the tool and a defect-free product. This analysis was carried out on a commercially available finite element package with appropriate nonlinear material and friction model.

Many investigations showed that numerical models could help in a better understanding of the forming procedure and also showed good correlations with the experimental results. Fu and Li (2009) have presented 3D-FE simulations and investigated the deformation behavior of the flexible die-forming process. The comparison between the conventional deep drawing and the visco-plastic carrying medium based on flexible die forming was conducted in terms of wall thickness reduction, hydrostatic pressure, principle stress distribution, and damage factor. The concave and convex rubber-pad forming process using FE simulations and experimental methods was investigated by Liu *et al*. (2010). The investigations of the forming load, thickness variation of the formed plate, and variations in the channel width to rib width ratio were also performed. Fabrication of a metallic bipolar plate for proton membrane in fuel cells is presented by Liu and Hua (2010). The FE analyses were used to describe the rubber-pad forming process and to investigate main parameters (such as rubber hardness and dimensions of the rigid die). It was found that in Figure (1): a) geometrical model of rib with a lightening hole, and b) geometry parameters used in FE simulations of rubber pad forming in a rib with a lightening hole smaller internal radius is, the harder is to fill the cavity of rigid die. The authors examined whether the blank filled the cavity of the rigid die or not by using a 3D laser scanning measurements system. In this case, numerical simulations of the rubber-pad forming processes were to analyze the blank and rubber behavior during the production of supporting ribs (rib with a lightning hole), as well as to analyze different tool geometry (Fig. 1). To find the optimal tool shape, a Non-linear FE analysis was conducted to predict the stress and strain distributions and the forming forces during the rubber-pad forming process. So, the main goal was to develop a computer model that would be able to simulate the process and, therefore, achieve the right design for a tooling set.

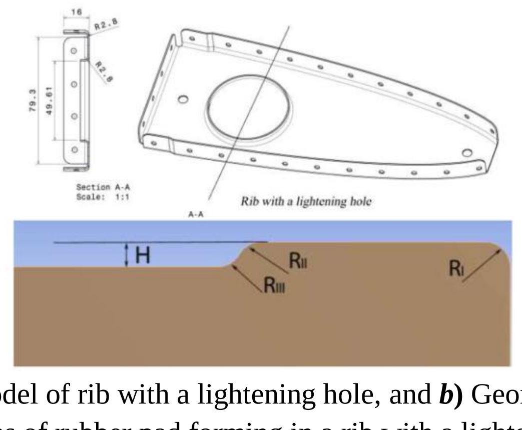

Figure 1. *a*) Geometrical model of rib with a lightening hole, and ***b*)** Geometry parameters used in FE simulations of rubber pad forming in a rib with a lightening hole

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

2. Numerical Simulation

Numerical simulations of the rubber-pad forming processes are complicated, mainly because of the large deformation of the rubber-pad. Consequently, a mesh distortion may occur in the simulation, which could lead to inaccurate and incomplete results. This is the reason why FE analyses must be carried out carefully and with a well understanding of the physical phenomena of the rubber pad forming process.

The commercial finite element software ANSYS was used to perform the FE simulation in this study. In order to reduce the processing time and improve the precision of calculations, 2D and 3D FE models were created. A rib with a lightening hole and analysis were carried out. The models in FE analysis have included three elements only: a rigid die, a blank and a rubber-pad (flexible punch). In order to simplify the numerical model, the container of the rubber-pad has not been modeled. Also in order to take into account the influence of the container, the frictionless support constraints were applied on opposite sides of the rubber, while displacement constraint was applied on upper edge of rubber model (Fig. 2).

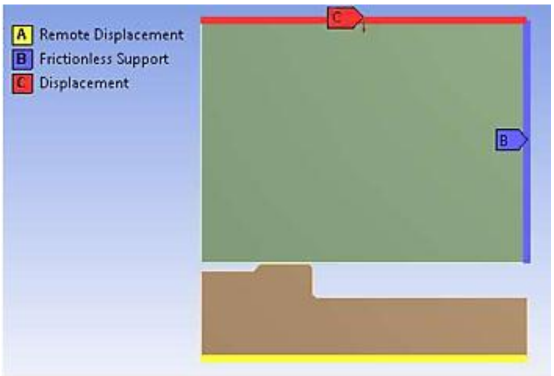

Figure 2. Constraints as applied in 2D symmetry FE model of rib with lighten hole**.**

The die was modeled as a rigid body because that the stress and strain of die were not analyzed and the die material (steel) is much less deformable than the material of the blank (aluminum). Hence, neither the material properties attached to the die were important, nor the mesh was generated. This procedure eliminated the unnecessary calculations that cause the decrease in both; run time and errors in the numerical solution.

All deformable materials in FE models have been modeled with Plane 183 finite element. The plane 183 has a quadratic displacement, plasticity, hyper-elasticity, creep, stress stiffening, large deflection and large strain simulation capabilities. However, in such models, die would be modeled as a rigid body, therefore, there would be no mesh generated for the die. The multi-linear isotropic hardening material which is existed in the ANSYS workbench and simulates a large plastic strain deformation is applied into the blank. Von Mises yield criterion coupled with an isotropic work hardening assumption were applied. The behavior of the nonlinear hyper-elastic and incompressible rubber-like material is described by Mooney-

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

Rivlin model. This model is based on the strain energy function and used for modeling the rubber pad. Two Mooney-Rivilin parameters C10 and C01 are used to describe the hyper-elastic rubber pad behavior. As it is mention above HD70 was used as a rubber pad, when the values of C10 and C01 are 0.736 MPa and 0.184 MPa respectively (Liu *et al*., 2010; Liu & Hua, 2010; and Dirikolu & Akdemir, 2004).

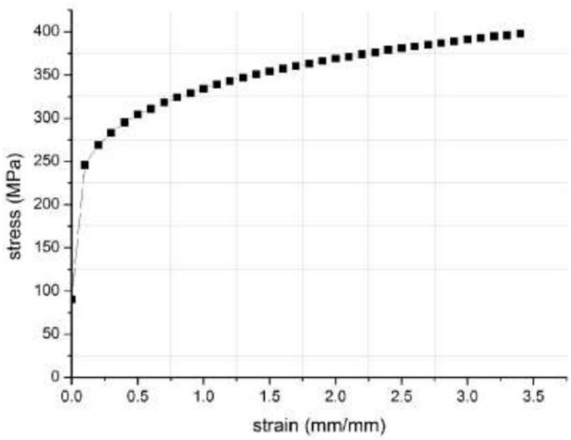

Figure 3. Experimental tensile-stress-strain curve for the aluminum blank sheet

In this work, an Aluminum alloy with a thickness of 0.6mm is used as a blank. The material properties of the blank were determined via stress stain curve obtained from the tensile test as shown in Figure (3). The elastic module (E) of this material is 71 GPa and Poisson ratio (ν) is 0.334. The frictional behavior between the rubber pad-blank and the die-blank are assumed to be following the coulombs model. The friction coefficient at the former and later contact pair was considered to be 0.2 and 0.1 respectively (Liu *et al*., 2010; Liu & Hua, 2010; and Dirikolu & Akdemir, 2004).

3. Results and Discussion

Figure (4) shows that the step by step forming process of deformation supporting rib with a lightening hole as it is mentioned above. Figure (4) illustrates that the forming process could be divided into three different stages: the first one is the self-deformation of the rubber pad, the second stage is the outer bending forming and the final one is the blank flows into the cavity of the die. During the forming process, thinning and thickening phenomena might occur. However, if the maximum thickness reduction reaches a critical value, the part would crack. In real production, this phenomenon should be avoided. According to Benisa *et al*. (2014 & 2012) and Sala (2001) the maximum reduction of this alloy is 20% which means that by using the safety factor as 1.2, the maximum plastic strain will be 0.223. Therefore, the plastic strain will be 0.186. The value of (0.186) for the plastic strain was used as a limit of FE simulation models of the supporting rib with a lightening hole. Furthermore, according to Takuda and Hatta (1998), there was a fracture initiation in the tensile specimen with no

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

obvious necking phenomenon, despite that; the alloy sheet has a considerable high work-hardening exponent of 0.19 where the elongation was only 17 pct (0.16 in true strain).

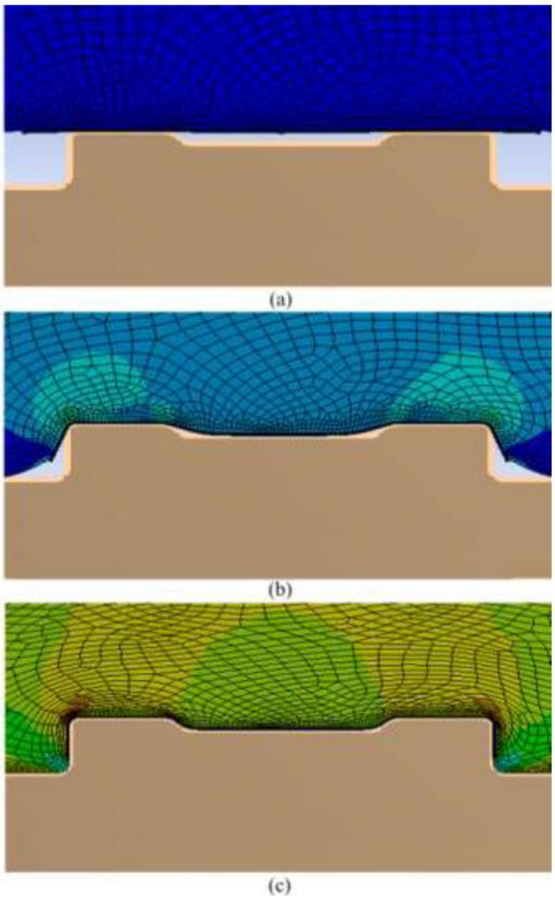

Figure 4. Three stages of supporting rib forming using rubber pad forming process.

In this study, the model of the rib with lightening hole (Fig. 1a), which has reached 0.186 of the plastic strain is unacceptable model. Generally, some parameters have important factors during the rubber pad forming process such as; the tool geometry, the hardness of the rubber and the lubrications. The tool geometry which is presented here as R_I, R_{II}, R_{III} and H (Fig. 1b) parameters has been studied using the rubber pad forming process. In order to study the effect of the previous geometry parameters values during the simulation of the rubber pad forming process, these parameters have been varied separately by fixing the first three parameters at 2 mm and changing the fourth one. This procedure was repeated with the rest of the parameters respectively to figure out the influence of each one of them (three are fixed, one is varied). In order to find the connections between the values of the fillet radius and the plastic strain and on the basis of these findings, more FE models of the tool with different values of fillets´ radii (R_I, R_{II}, R_{III}, and H) have been developed and analyzed. These simulations showed that, the values of the stress and the strain are strongly dependent on the rib geometry. Different models based on different dimensions were simulated. R_I was varied from 1 to 5 mm, and the other parameters (R_{II}, R_{III}, and H) were varied from 1 to 5 mm.

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

Figure (5) illustrates the plastic strain and the geometry parameters relationship for each model. As it is seen in figure 5 the plastic strain is strongly dependent on the geometry parameters (R_I, R_{II}, R_{III}, and H). It is also clear that the increasing of R_I, R_{II}, R_{III} and the decreasing of H leads to the plastic strain decrease. On the other hand, the capability of forming the blank increases. However, when R_{II} and $R_I \geq 2$ and $H < R_{II}$ the plastic strain was < 0.186 mm/mm (Fig. 5). In the case of the R_I 1.5 mm, the plastic strain is greater than the acceptable value where it reached the 0.211, and the stress concentrated at the radius R_I region was 266.4 MPa. The reason of that might be, according to reference (Thiruvarudchelvan, 1993), where it mentioned that, the blank could be suffer affected not only by tensile stress and tangential stress but also from the stress resulted from the bending pressure imposed by the tool. When the $R_I \geq 2$ (R_I=2.5mm), the stress and the plastic stain started to decrease until they reached the 243.9 MPa and 0.15 mm/mm values respectively (Fig. 6). The same attitude is seen for $R_{II} \geq 2$ as shown in Figure (7) in where two values of R_{II} were selected (1 & 3 mm).

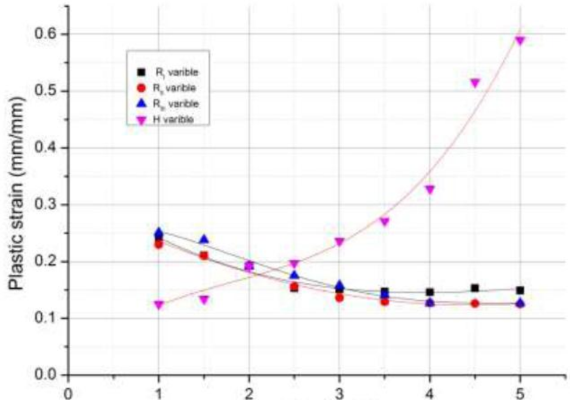

Figure 5. Influence the fillet radius of the rib on plastic strain

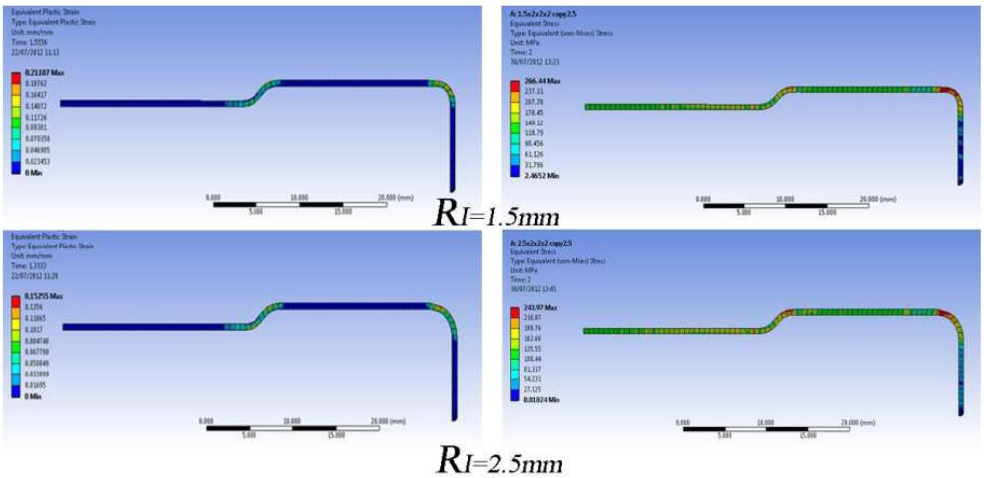

Figure 6. Plastic strain (left column), Equivalent stress (right column).

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

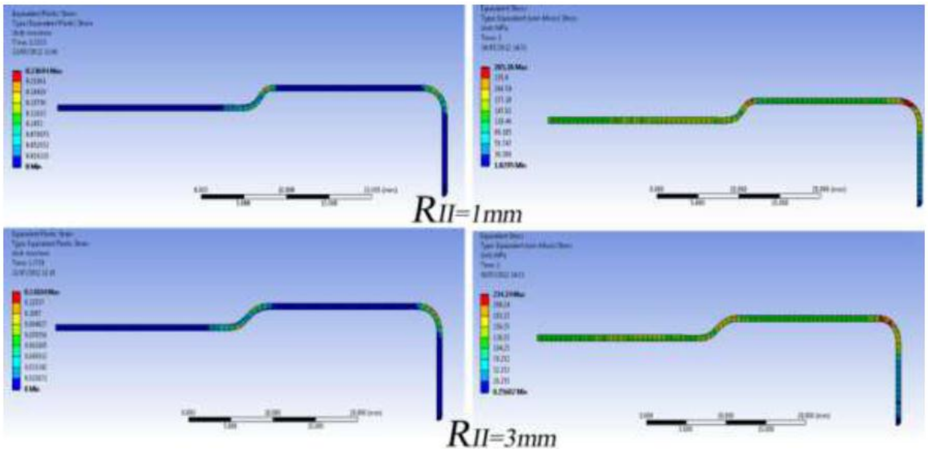

Figure 7. Plastic strain (left column), and Equivalent stress (right column).

As mentioned previously, the forming force was presented as a displacement applied on the upper side of the aluminum billet (Fig. 2). The convergences of the forming forces for ECAP model, obtained through the FE simulations, are shown in Figure (5), this figure shows that the highest value of a forming force is (37,833 N). It can be seen that the magnitude of the forming force increases as the simulation time increases (as the punch extruded the billet through the inter-sectioning channel).

According to Figure (8), the R_{III} values in all models should be greater than or equal to R_{II} in order to make an easy forming and to avoid uncompleted cavity tool forming. Also from Figure (8), when R_{III} value is less than the value of R_{II} (equal to 1.5 mm) the die cavity was uncompleted filling with higher plastic (0.251 mm/mm). On the other hand, at increasing the value of R_{III} up to 3.5 mm the die cavity was completely fill with low plastic strain (0.142 mm/mm). H value has a strong influence upon the plastic strain in the blank. In the case where $H \leq R_{II}$ (H=1.5 mm), the plastic strain value is always less than 0.186 while it reaches 0.143 mm/mm when the stress is 269.5 MPa. Otherwise, the plastic strain would be increased higher than the acceptable plastic strain (0.4378 mm/mm) for H= 4.5mm (Fig. 9). Moreover, when these parameters were randomly selected, we got the same results that have been reached in the previous case. Such models (the randomly selected) for $R_{II} > 2$ and $R_I \geq 2$, all models have a plastic strain within the limit. As stated earlier if, in all models, $R_{III} < R_{II}$ then the cavity filling will be uncompleted. On the other hand, the plastic strain exceeds the reference plastic strain (0.186) when $H > R_{II}$.

To get a plastic strain less than 0.186 in the blank and a complete filled cavity of the tool, the values of R_{II} & R_{III} should be increased and the H value decreased. As it is known, increasing the bending radius leads to the undesirable increase of the spring back. Therefore, in order to get an acceptable tool design, we have to make some kind of a compromise of these parameters values.

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

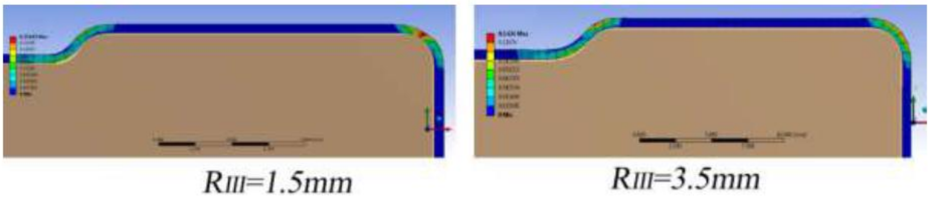

Figure 8. Effect the RIII on the filling the die cavity.

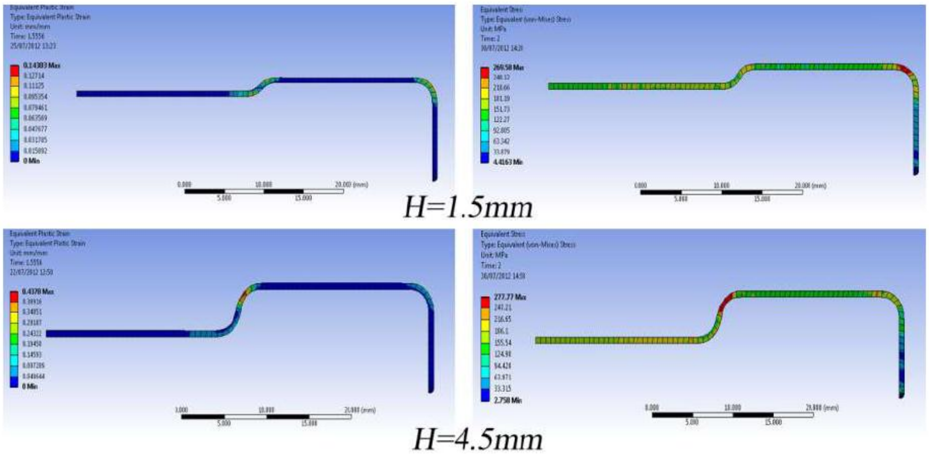

Figure 9. Plastic strain (left column) Equivalent stress (right column).

In addition, after obtaining satisfactory results in 2D simulations, a decision to perform more complex and challenging 3D simulations of the same process has been made. In ANSYS the 3D model definition (which is almost as the same as the model used for the 2D analysis), the die has been defined as a rigid body, while the rubber pad and the work piece were represented by a deformable mesh. The friction coefficient, material properties and constrains used in the 3D simulation were taken from the earlier mentioned 2D rubber bad forming simulation (Fig. 2 for the material properties and Fig. 3 for the constrains). Figure (10) shows the complete 3D FE model as used in the simulation. It can be seen that one-half of the real set-up was modeled and then symmetry conditions were applied.

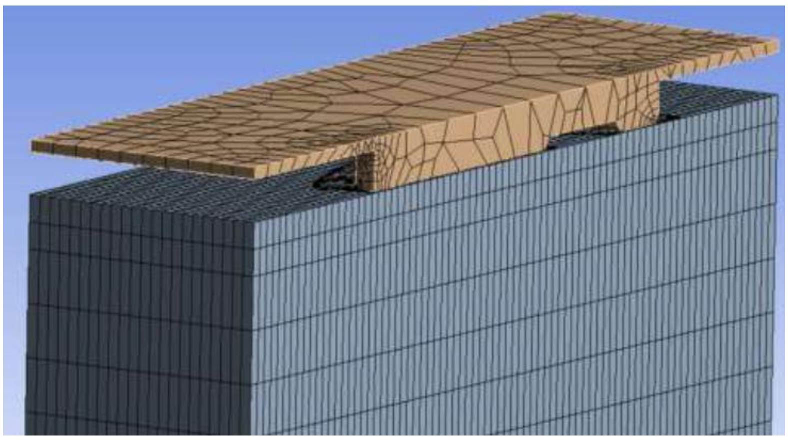

Figure10. FE mesh used in the analysis of symmetrical model of the rib forming using a rubber pad.

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

In FEM simulations, in the 2D simulations of the sheet metal bending, we cannot see the wrinkling in the sheet metal. Hence, this is why the 3D simulations have been used, where they allow illustrating the wrinkling region in the rib during the rubber pad forming process.

Figure (11) shows the 3D symmetry model of the formed rib with lightening hole as obtained from the simulation. There is no wrinkling in the straight flange, where it occurs in the curved flange and increases as the curvature of the flange increases. This has been verified through the experiment as it can be seen at the right hand side of Figure (11).

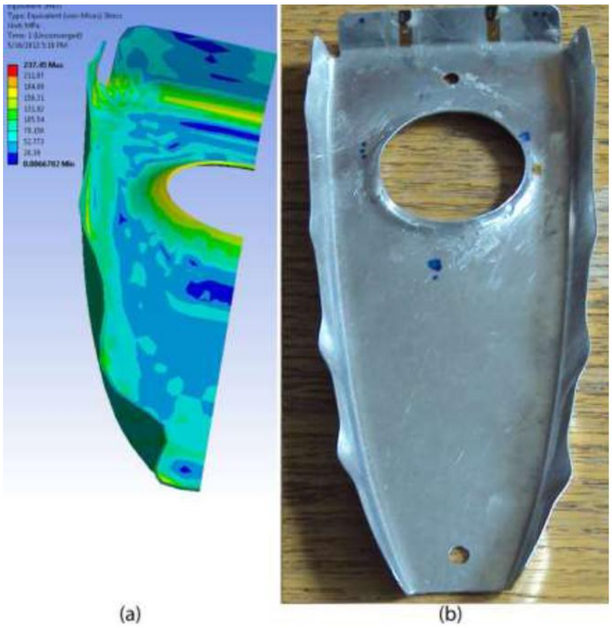

Figure 11. Wrinkling in FEM model of formed rib with lightening hole.

4. Conclusion

A finite-element simulation of the rubber-pad forming process could be a very useful tool for understanding and improving the forming operations, where it provides important data for determining the forming parameters and the operation time. The developed FE models and the method proposed in this paper have been proven to be sufficiently effective in predicting the final shape of the component and the regions of a possible crack appearance. The FE simulations showed that the maximum stresses and strains in all cases were at the flanges and the corners. The minimum stress and plastic strain were achieved in the straight rib (the rib with the simplest geometry), while the maximum stress and plastic strain were found in the rib with a lightning hole (the most complex geometry). These results have been validated by the experiments, as well as by the fracture criterion used for the crack predictions. The FE simulations have proved that simpler tools would reduce the lead times and enable the rapid production of small parts without a possibility of a crack appearing during forming. On the other hand, the geometry of more complicated, but necessary, tools must be defined very carefully, with the determination of the fillet radii that will minimize the chance of a fracture. FEM can help us with this determination too, while additional potential applications – such as

ISSN (Print): 2413-5267
ISSN (Online): 2706-9966

3D model simulations and tool optimization – are also possible applications. However, it must be noticed that the optimization procedure of the press-forming processes – owing to the presence of the hardly reproducible phenomena like friction and lubrication – should never be limited to simple numerical simulations, because the above phenomena can contribute a lot towards saving the costs and reducing the time-to-market, currently held up by empirical trial-and-error processes. Sheet metal-forming-simulation results, today, are reliable and accurate enough so that even the try-out tools and the time-consuming try-out processes might be eliminated or at least reduced significantly.

References

ASM (2006). *ASM Handbook Vol.14B, Metal Working: Sheet Forming*. ASM International, pp. 375-385.

Benisa M., Babic B., Grbovic A., and Stefanovic Z. (2014). Numerical Simulation as a Tool for Optimizing Tool Geometry for Rubber Pad Forming Process. *FME Transactions*, 42: 67-73.

Benisa M., Babic B., Grbovic A., Stefanovic Z. (2012). Computer-aided modelling of the rubber-pad forming process. *Materials and Technology*, 46(5): 503–510.

Dirikolu M.H. and Akdemir E. (2004). Computer aided modelling of flexible forming process. *Journal of Materials Processing Technology,* 148(3): 376-381.

Fu M.W., Li H., Lu J., & Lu S.Q. (2009). Numerical study on the deformation behaviors of the flexible die forming by using viscoplastic pressure-carrying medium. *Computational Materials Science*, 46(4): 1058-1068.

Liu Y. and Hua L. (2010). Fabrication of metallic bipolar plate for proton exchange membrane fuel cells by rubber pad forming. *Journal of Power Sources*, 195(11): 3529-3535.

Liu Y., Hua L., Lan J., & Wei X. (2010). Studies of the deformation styles of the rubber-pad forming process used for manufacturing metallic bipolar plates. *Journal of Power Sources*, 195(24): 8177-8184.

Sala G. (2001). A numerical and experimental approach to optimize sheet stamping technologies: Part II - aluminum alloys rubber-forming. *Material and Design*, 22: 299–315.

Takuda H. and Hatta N. (1998). Numerical analysis of the formability of an aluminum 2024 alloy sheet and its laminates with steel sheets. *Metallurgical and Materials Transactions A*, 29: 2829-2834.

Thiruvarudchelvan S. (1993). Elastomers in metal forming: a review. *Journal of Materials Processing Technology*, 39(1-2): 55-82.