IOP Conf. Series: Materials Science and Engineering **212** (2017) 012003 doi:10.1088/1757-899X/212/1/012003

Investigation of contact pressure in auto panel drawing dies during the forming process by using FEM method

FatihKoşar, Kemal Yaman

Depth. Of Mechanical Systems, Research and Development Institute, Turkish Scientific and Technical Research Council, (TÜBİTAK-SAGE), 06261, Ankara, Turkiye

E-mail: fatih.kosar@tubitak.gov.tr; kemal.yaman@tubitak.gov.tr

Abstract. In automotive stamping dies, the die design is generally based on experience. Usually, the loads on the die is not considered too much when the ribs, which strengthen the structure of the die, are positioned. Despite an abundance of manuscripts in the literature on formability and springback, the number of studies that examine the panel-die interaction is rather limited. The impact of the loads occurs in the form of contact pressure (CP) on the upper and lower die when the sheet metal is formed in a drawing die. For die designs, the CPs on the die must be calculated accurately in order to determine the correct position for the ribs and to optimize the weight of the die. The present study compares the CP distribution on a auto panel drawing die through different solution methods to examine the time-dependent change of the CP.

1. Introduction

Despite there are many studies about panel quality in the forming process, the number of studies on the metal-die interaction is somewhat sparse. In a die design, the positioning of the ribs is based on the experience and foresight of the designer. Although the weight of the vehicles has gradually decreased, no changes have occurred in the weights of the dies. The sole reason for this is because companies and academia focus more on panel quality, while die-panel interaction is not sufficiently investigated. However, the weight of stamping dies, as high as 40 tons, can be significantly reduced through optimization. This will allow cheaper dies and a larger variety of models to be offered to customers.
The interaction between metal and the die occurs in form of contact pressure (CP). In order to ascertain the stress occurring in the die during the forming process, it is first necessary to correctly determine these CPs. Nowadays, finite elements software can be used to easily determine the CPs.The objective of the present study is to determine the loads affecting a draw die in the form of CP and to examine the stresses created by such loads on the die. By inspecting the stress values on the die, conventional die design methodology can be questioned whether over-design or not. Due to low strain rates, the forming process is accepted to be quasi-static [1-5]; however, during forming process the CP values could change drastically. In the present study, the CP obtained through the quasi-static solution is compared with those obtained through the time-dependent solution.

There are some studies in literature about the effect of CP on die wear and tool life. Pereira et al. investigate the distribution of CP over the die radius, throughout the duration of channel forming

IOP Conf. Series: Materials Science and Engineering **212** (2017) 012003 doi:10.1088/1757-899X/212/1/012003

process for estimation of tool wear and life. Their study reveals that the CP response can be divided into two distinct phases that are steady state and transient phases. In steady state phase, CP remains almost constant however, in transient phase CP is more than double the magnitude of the steady state peak pressure. They also compared the quasi-static and transient FE simulation results for maximum CP distribution on die radius during the both phases. Quasi-static simulation results showed good correlation with the transient simulation results for both the transient and steady state regions [6]. In another study, Pereira et al. directly examines the contact sliding distance experienced during a typical sheet metal stamping process. Analysis of the contact conditions experienced by the blank showed that there is a region on the blank surface that experiences longer sliding distances at very high contact pressures. The analysis of the conditions at the die and blank surfaces both showed that the identified transient stage is likely to be critical to the overall wear/galling behavior [7-8].
A new slider type of test system was developed by Cora et al. [9] in order to replicate the actual stamping conditions including the CP state, sliding velocity level and continuous and fresh contact pairs (blank-die surfaces). Test results showed that the TD and CVD coatings have performed better wear resistance performance on contact surfaces.

2. Numerical Study

In this study, two different finite elements software were used to determine and compare the CPs. One of the software used is Autoform®, a program that specializes in metal forming, that is frequently used in the automotive sector, whereas the other is Abaqus®, which has strong modelling and solutions for non-linear analyses. The implicit method is used for the Autoform solution method and the explicit method is preferred for the Abaqus solution; thus, the outcomes of the quasi-static and time-dependent analyses of the CP were compared. In die analyzes the ANSA® software, which has the ability to create high convergence mesh network is used and for post processing the MetaPost® is used.

2.1. Form analysis

The forming analysis model of Abaqus explicit is shown in Figure 1 and Figure 2. All die elements, except the sheet metal, were accepted as rigid. The 3D CAD data of the die in the form analysis were obtained from a design, which were manufactured before.

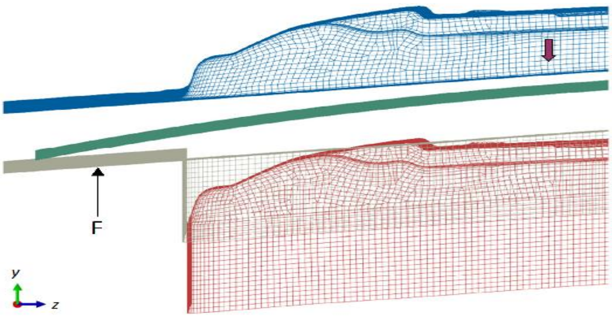

Figure 1. Form analysis model

The lightening holes on the upper die, which can be seen in the Figure 2 also included to forming analysis.

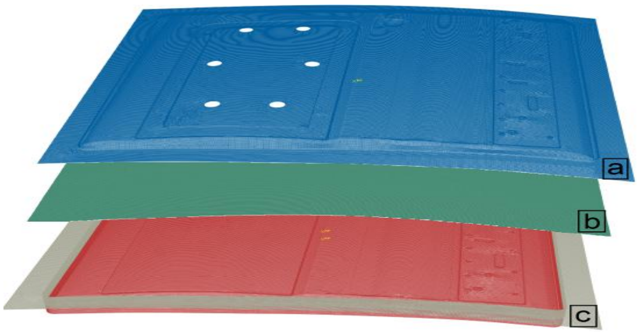

Figure 2. (a) Upper die, (b) sheet metal, (c) lower die and blank holder surfaces

The analysis model was built to allow for the sheet metal thickness (0.65 mm) when the model is in full-closed status. (See Fig. 2)

2.2. Model parameters

Some of the parameters and their properties that directly affect the analysis results are listed below:
Blank (Panel) Material: Material tensile test were performed for true stress-strain relationship of blank material. The Figure 3 shows the plastic hardening curve of blank material (bake hardenable steel CR240B2) obtained from the test.

Table 1. Properties of die material [10-11]

Die Material	Yield Stress [MPa]	Ultimate Stress [MPa]	Modulus of Elasticity [GPa]	Poisson ratio
EN-JS 2070	420	700	170	0.26

Die Material: Die material used in the analysis is EN-JS 2070 (GGG 70L) cast iron. The properties of die material are given in Table 1.
Total Time: Since the critical time increment of explicit analyses is very small, the total solution time may be longer when compared with the implicit methods. In forming analysis, total analysis time can be shortened by increasing the die speed far above the actual conditions, but artificial dynamic effects that cannot occur in the real world can occur. In metal forming analyses, a parameter is used to avoid such effects. These parameters are the explicit time step required for 1 mm movement of the die (i.e. ncpm: number of cycles per millimeter). A value between 100-1000 is recommended [12]. For the present study, this value was set to 100. Also, the maximum die speed recommended for explicit analyses is 2 mm/ms.
Speed: For the analyses, the upper die speed was defined as linear with first 2 ms and last 2 ms so that the first and the last speeds are both 0, hence reducing the dynamic factors.

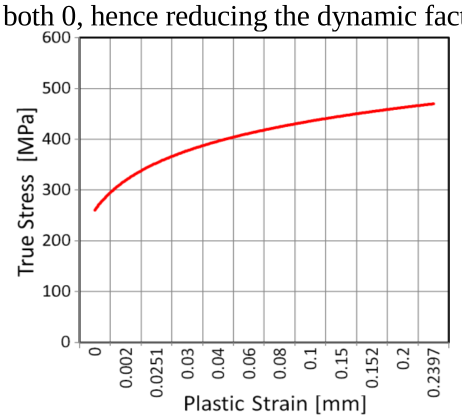

Figure 3. True stress-strain curve of blank material

Mesh: Since the element size needs to reflect the actual geometry and forms, an average 4 mm was chosen for lower die, upper die and blank holder and 3 mm was chosen for the sheet metal. S4R (shell quadrilateral reduced integration) was the element type selected for the metal sheet.
Contact Characteristics: For contact definition, isotropic penalty friction formulation was used. 0.15 friction coefficient was used in simulations. For contact stiffness, a "hard contact" pressure over-closure relationship is used.
Boundary Conditions: Lower Die rigid body reference point is fixed and a speed profile was assigned to the upper die.
Force: For the blank holder pressure force, which is one of the most important parameters of the form analysis, a force value of 2300 kN was used as provided by the Autoform. The gravitational acceleration was also included in the analysis.

3. Result and discussion

Figure 4 shows the maximum CPs obtained by scanning the time intervals in the Abaqus Explicit analysis and the maximum CP gained from Autoform solution for sample forms.

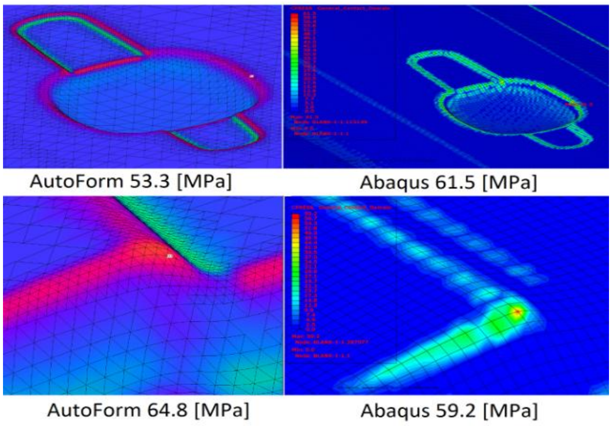

Figure 4.Abaqus/Autoform contact pressures comparison

When the Autoform and Abaqus CPs are compared, it is seen that zero CP occurs in flat sections of panel in both analyses. At local forms a difference of up to %15 are seen between the two results. The solution method (implicit/explicit), algorithms and mesh differences are the main reasons for the difference. CP change during the forming process for a "S" shape form can be seen in Fig. 5.

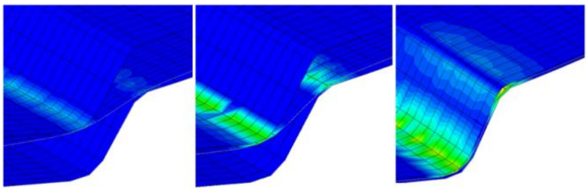

Figure 5. Contact pressure change during the forming process

Points labeled on the die surface as seen in Figure 6a. The time-dependent CP change of the points on this form are given graphically in Figure 6b.

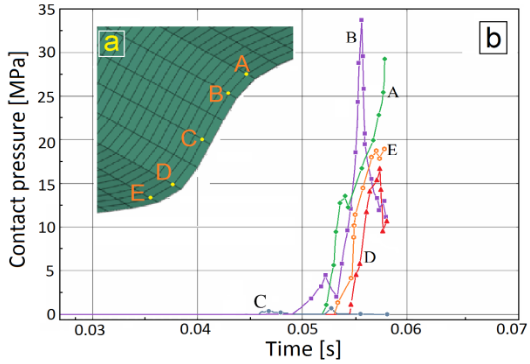

Figure 6. The time-dependent pressure change of the points (a) labeled on the die surface and (b) graphical results

Although high CPs are seen locally in the analysis of Autoform and Abaqus, the average value of both outcomes is concentrated around 60 [MPa]. In order to investigate the effect of CP on the lower die, a static analysis was performed using the maximum CP values obtained from the previous Autoform results. The panel CP values shown in Fig. 7a were applied to the relevant area on the lower die.

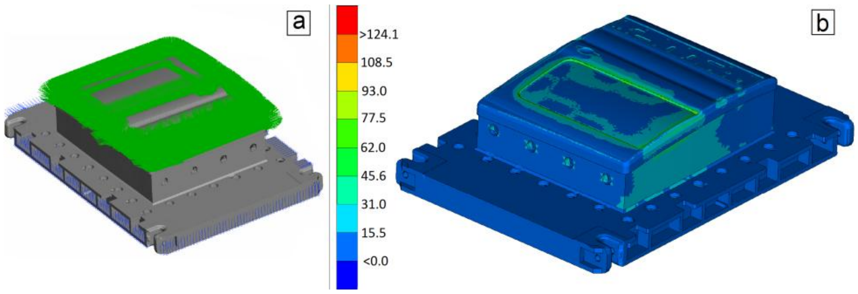

Figure 7. Boundary conditions of lower die for CP analysis (a) and CP analysis result on lower die (b)

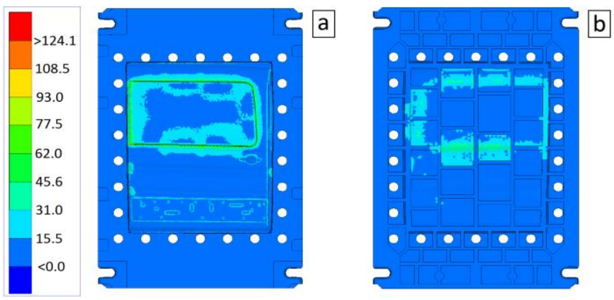

Figure 8. Result of CP analysis on lower die (a) top view (b) bottom view

The results of static analysis on lower die is shown in Fig.7b Stress distribution over the die geometry is also given in top and bottom view in Fig. 8a and Fig. 8b respectively. The highest stress values were formed on the surface of glass zone in parallel with the CP form analysis values.

4. Conclusion

In the present study, the CP obtained from Autoform Implicit and Abaqus Explicit solution method are compared. This comparison showed that the time-dependent and quasi-static CPs were almost similar. It was seen that the similar CP results obtained through different solution methods can be used for die-blank (panel) interaction. The stress results due to the obtained maximum CPs can be used for die fatigue analyses and die optimization studies. Furthermore, when the CP results are examined, it is seen that the forming process occurs through the male and female forms in the lower and upper die.

With this approach, the flat sections need not be pressed between the lower and the upper dies. In fact, in the Abaqus form analysis, the lightening holes located in the upper flat area were added to the calculations and there was no adverse effect on the panel quality of these sections. If the flat sections are removed on the lower and upper die, savings can be made in terms of material, machining and the heat treatment. As a result, for die design, the CPs on the die must be calculated accurately in order to determine the correct position for the ribs and to optimize the weight of the die.

Acknowledgments

The authors wish to thank to the FORD Otomativ San. A.Ş. Tool&Die Depth.for their motivations, supports of equipment and workforce in order to carry out the study.

References

[1] ABAQUS Inc. ABAQUS Version 6.14 Documentation, Dassault Systems, Providence, RI 029009, 2004.

[2] N. Rebelo, J.C. Nagtegaal, H.D. Hibbit, “Finite element analysis of sheet forming process”, Int. j. for Num. Methods in Eng., vol. 30, pp.1739-1758, 1990.

[3] S.C. Tang, “Trends on simulation of sheet metal forming process”, SAE Transactions: J. of Mater. & Manuf. Vol. 109, pp. 523-530, 2000.

[4] A.E. Tekkaya, “State-of-the-art of simulation of sheet metal forming”, J. of Mater. Proc. Tech., vol. 103, pp. 14-22, 2000.

[5] M. Tisza, “Numerical modelling and simulation in sheet metal forming” J. of Mater. Proc. Tech., vol. 151, pp. 58-62, 2004.

[6] M.P. Pereira, W. Yan, B.F. Rolfe, “Contact pressure evolution and its relation to wear in sheet metal forming” Wear, vol. 265, pp. 1687-1699, 2008.

[7] M.P. Pereira, W. Yan, B.F. Rolfe, “Sliding distance, contact pressure and wear in sheet metal stamping” Wear, vol. 268, pp. 1275-1284, 2010.

[8] M.P. Pereira, W. Yan, B.F. Rolfe, “Wear at the die radius in sheet metal stamping” Wear, vol. 274-275, pp. 355-367, 2012.

[9] Ö.N. Cora, K. Namiki, M. Koç, “Wear performance assessment of alternative stamping die materials utilizing a novel test system” Wear, vol. 267, pp. 1123-1129, 2009.

[10] W. Callister, “Fundamentals of Materials Science and Engineering”, 5th John Wiley & Sons, Utah, 2000.

[11] K.M. Zwilsky, “Properties and Selection Irons Steels and Performance Alloys”, Vol.1 2nd ed., ASM, U.S.A, 1990.

[12] B.N. Maker, X. Zhu, “Input parameter for metal forming simulation using Ls-Dyna”, Liwermore Soft. Tech. Co., 2000.