J MATER RES TECHNOL. 2014;3(1):71–78

Available online at www.sciencedirect.com

jmr&t
Journal of Materials Research and Technology
www.jmrt.com.br

Original Article

Finite element simulation of ironing process under warm conditions

Swadesh Kumar Singh[a,*], Vinay Kumar[a], Paresi Prudvi Reddy[a], Amit Kumar Gupta[b]

[a] Department of Mechanical Engineering, Gokaraju Rangaraju Institute of Engineering and Technology, Bachupally, Hyderabad, India
[b] Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, Hyderabad, India

ARTICLE INFO

Article history:
Received 19 August 2013
Accepted 24 October 2013
Available online 13 December 2013

Keywords:
Deep drawing
Ironing
Warm forming
Finite element simulation

ABSTRACT

Metal forming is one of the most important steps in manufacturing of a large variety of products. Ironing in deep drawing is done by adjusting the clearance between the punch and the die and allow the material flow over the punch. In the present investigation effect of extent of ironing behavior on the characteristics of the product like thickness distribution with respect to temperature was studied. With the help of finite element simulation using explicit finite element code LS-DYNA the stress in the drawn cup were predicted in the drawn cup. To increase the accuracy in the simulation process, numbers of integration points were increased in the thickness direction and it was found that there is very close prediction of finite element results to that of experimental ones.

1. Introduction

In conventional deep drawing, the drawn cups will have thicker walls at its rim than at its base if the clearance is larger than sheet thickness. The process is used for various industrial applications including manufacturing of beverage can, air filter, etc. Ironing is a process in which the wall thickness of a drawn cup is made uniform by the pushing of the cup through reduced clearance or ironing rings [1]. Further, residual stresses are induced during deep drawing affecting fatigue strength of the part, promoting stress crack corrosion, or resulting in dimensional changes after the machining of the formed parts. Ironing can be used to reduce the residual stresses in deep drawn cups apart from imparting improved geometrical accuracy [2]. In a deep drawn cup tensile stresses are formed over the outer surface and compressive stresses on the inner surface. Ironing gives characteristic distribution of residual stresses along the cup wall outer surface [1]. Increasing the extent of ironing induces the compressive axial residual stresses over the entire wall outer surface which neutralizes the tensile residual stresses [3]. A FEM simulation of the ironing of a deep drawn cup has shown that residual stresses after ironing are reduced by 50–65% [4]. A large number of automobile components are manufactured by deep drawing followed by redrawing and ironing stages, e.g. the drum clutch is usually formed in 9 redrawing and 3 ironing stages [5]. A method of reducing these numbers of stages during forming and also to improve formability of material is warm forming. Parsa et al. [6] carried out rigid-plastic finite element analysis of the two-stage forward and reverse redrawing process. Baillet et al. [7] used an explicit finite element method in the analysis of the aluminum can ironing process.

Although the deep drawing process of high strength/low formability metals has an extensive industrial application area, deep drawing at room temperature has serious

* Corresponding author.
E-mail: swadeshsingh@griet.ac.in (S.K. Singh).
2238-7854 http://dx.doi.org/10.1016/j.jmrt.2013.10.013

difficulties because of the large amount of deformations revealed and high flow stresses of the materials [5]. Thus, crumples, wrinkles and earrings will occur on the product surface because of the anisotropy of the materials. Elevated temperatures decrease the flow stresses and increase the formability of the materials and thus deformations become easier. The effective basic mechanism in pressing is plastic deformation. Because of this, deformation temperature has to be determined by taking this point into consideration. The needed temperature for steel sheets is quite high and this situation should be the reason for the low number of studies about this material [6].

Chen et al. [8] investigated combined isothermal/non-isothermal finite element analysis (FEA) with design of experiments tools to predict appropriate warm forming temperature conditions for 5083-O (Al–Mg) sheet metal blanks, deep drawing and two-dimensional stamping cases. To achieve increased degrees of forming, different temperature levels should be assigned to the corner and body of the die and punch. 25–250% elongation ranges were seen. They found that the formability of Al-5083 alloy is greatly dependent on the temperature distribution of the die and punch. It is also observed that the optimal temperature distributions for warm deep drawing and warm two-dimensional stamping were not identical. In the present work, cylindrical deep drawing tests with reduced clearance (ironing) were performed on extra deep drawn (EDD) steel sheet of 2 mm thickness at various clearances between punch and die and at temperatures of 200–600 °C. Authors [9–13] in their earlier studies have carried out extensive studies in the warm forming of EDD steel. For this, low carbon material mechanical properties at various temperatures were investigated and using these properties the warm deep drawing process was simulated in explicit finite element code LS-DYNA. During extensive literature study on ironing it was found that in warm condition there is no work published so far and the present investigation was carried out to simulate the Ironing process in warm condition using Finite Element code LS-DYNA.

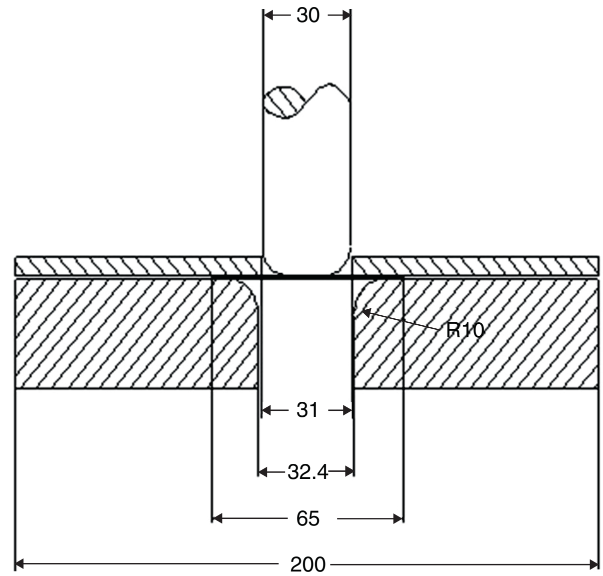

Fig. 1 – Tooling for 10% reduction in the thickness of 2 mm sheet.

2. Experimentation

The test specimens of different diameters were prepared on a lathe. These blanks were heated to a required temperature of 200 °C, 400 °C, etc. by using induction heater which was specially designed to heat materials up to 700 °C. But at high temperatures there is a tendency for the die material to expand. So the material for the dies should be such that it does not excessively change its dimensions otherwise the design of dies will change. The tooling material also should not lose its strength at higher temperatures. So, for this purpose Inconel-600 material was chosen to make dies. Another induction coil was attached around the lower die to heat it at a predetermined temperature to avoid thermal shock. The temperatures were recorded by using pyrometer which is a noncontact temperature detecting instrument. Since there will be an increase in the coefficient of friction by increasing the temperature, which may rupture the blank in the initial stages of drawing, a high temperature lubricant 'Molycote' was applied on the interface between die and deformable blank and blank and blank holder. This lubricant becomes especially effective at higher temperatures. Tests were conducted on two sets of dies to provide different extent of ironing. For 10% and 25% reduction dies are shown in Figs. 1 and 2. The photographs of some of these cups are shown in Fig. 3. Most of these results were studied and analyzed by the authors in their earlier study [10].

3. Finite element simulation

As discussed in the previous section, finite element method has been extensively used in forming operations to optimize various process variables in order to produce defect-free parts. Generally, in any operation large amount of time is consumed in trial and error method and there are high chances that the tools are to be redesigned whenever the desired products are not obtained. So, this trial and error method involves a lot of expenditure and loss of valuable time. To overcome this problem, process modeling by computer simulation called Finite Element Method (FEM) has been introduced which simulates the actual process and thus saves time and money. Many commercial codes are available for Finite Element Analysis in metal forming such as Dynaform, Abacus, Nike 2D, etc.

The finite element analysis is done using a commercially available code Dynaform version 5.6.1 with LS-Dyna version 971 solver with coupled thermal analysis for deep drawing at elevated temperature. The code was developed for applications such as sheet metal forming, automobile crashworthiness, occupant safety and underwater explosions. It is a non-linear dynamic simulation package which can simulate different types of sheet metal processes including deep drawing, stretching, bending, hydro forming, stamping, etc. to predict stresses, strains, thickness distribution, etc. and the effect of various design parameters of tooling on final product can be studied.