ELSEVIER

Journal of Materials Processing Technology 172 (2006) 139–145

Journal of Materials Processing Technology

www.elsevier.com/locate/jmatprotec

Experimental evaluation and finite element simulation of explosive forming of a square cup from a brass plate assisted by a lead plug

V.N. Wijayathunga *, D.C. Webb

School of Engineering, Leeds Metropolitan University, Calverley Street, Leeds LS1 3HE, UK

Received 2 September 2002; received in revised form 20 March 2003; accepted 5 September 2005

Abstract

A FE model is developed to simulate the experimental tests for the impulsive deep drawing of a brass square cup with the presence of a lead plug. The loading of the assembly is achieved by the detonation of a high explosive, underwater. The formability of the brass plate in the absence of the lead plug gives rise to material instability and uneven material thickness over the formed region causing rupture. The presence of lead plug initially slows down the forming process by acting as a reservoir for the kinetic energy. As the deformation progresses, kinetic energy in the plug is transferred to the plate and the increased time scale of the operation enables the system to be treated as a quasi static forming process. In addition to postponing the onset of material instability, the presence of the lead plug enables a higher ratio of drawability and a better uniformity in thickness of the formed dome. The experimental investigations have indicated that the optimum shape of the plate specimen is a square blank with circular cut-off segments. The effects of the medium impedance, wave reflection and refraction are considered to be negligible in order to improve the simplicity of the modelling procedure. Finally, the computed results are compared with those obtained from experimental tests.

Keywords: Explosive forming; Square cups; Impulse loading; Stand-off distance; Plug cushion

1. Introduction

In recent years, the application of sheet metal parts in production has become more frequent. Scientific, numerical and engineering advances in sheet forming operations are made at a rapid rate. Hence, accurate and efficient simulations of these operations have become significantly important. There are many investigations pertaining to axi-symmetric and square cup drawing operation with reference to formability limit. A representative example could be found in references [1–5]. In all these cases the sheet metal is stretched and formed in to required shape. Explosive forming is a technique that is generally adopted for sheet metal forming operations that involve very large sheets and usually axi-symmetric. Description of the method and its applications could be found in references [6–8]. The method is essentially dynamic and the forming is induced by exposing sheet metal surface to an incoming pressure wave generated by underwater explosion. But the direct application of the pressure wave on the metal surface gives rise to a stress–strain field distribution of unacceptable profile, which may lead to premature failure of material. Hence, to improve the formability limit an alternative technique referred to as ‘plug cushion forming’ has been introduced [9,10]. A plug of soft material is inserted in contact with plate, in between the source of explosion and the plate. Thus, the pressure wave instead reaching the plate directly exerts itself on the plate via the plug. Depending on the thickness of plate and plug as well as the angle of the plug chamfer, the deformation process is slowed down. Since the plug is made of a relatively soft material, its relative energy absorption capacity is low. This allows it to act as a conduit to transfer the energy at a relatively slower rate to the plate. Experiments conducted for brass plate specimen and lead plug are specified in reference [10]. The measured thickness strain is compared with numerically simulated results produced using ABAQUS FE Explicit code version 6.2 [11] and the compatibility between the two is found to be acceptable.

2. Materials, procedure and equipment

2.1. Blank material

18 SWG (1.2192 mm) brass squares of 203.2 mm (8 in.) sides with corners machined circular to 215.9 mm (8.5 in.) diameter (see Fig. 1). The static stress–strain curve for this material obtained for the ‘as received’ condition employing the bulge test is given in Fig. 5.

* Corresponding author.

0924-0136/$ – see front matter
doi:10.1016/j.jmatprotec.2005.09.014

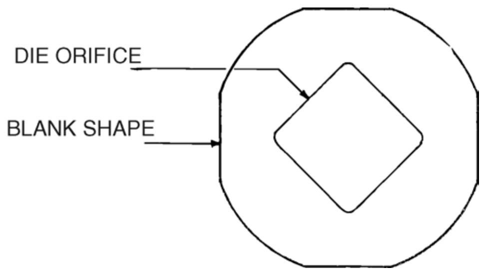

Fig. 1. Blank.

2.2. Explosive materials

The explosive used in the experimental investigations is a plastic sheet explosive known commercially as ‘Metabel’. This explosive material is supplied in the form of sheets 250 mm × 125 mm and 3.175 mm thick, and it is mouldable and has good water resistance. The calorific value of the explosive is not known but its detonation velocity is 6.7 km/s and its density 1.53 g/cm^3.

2.3. Die details

The free forming of dome-shaped square cups is achieved using a die with square ducts of 101.6 and 88.9 mm square, a corner radius of 3.175 mm and an entrance profile radius of 12.7 mm (see Fig. 2).

2.4. Lead plug manufacture

A two-piece heated steel mould is used to produce 101.6 mm square castings, provided with pipes to ensure freedom from cavities. These castings are then machined into square plugs of 82.55 mm side (minimum) with up to a 45° chamfer angle and thickness of 12.7 mm.

3. Experimental procedure

A grid of measured concentric circles is printed on the blank, which is positioned and clamped as shown in Fig. 2. The explosive charge is placed centrally at a ‘stand-off’ distance of 300 mm. With the plug in position, the detonator and shot firing cable are attached and the whole assembly is lowered into the tank to the required hydrostatic head (610 mm). After obtaining the desired vacuum and the circuit checked with an ohmmeter, the exploder is connected and the charge detonated. Measurements of the grid positions are then retaken.

4. Experimental results

(a) The natural hoop strain, $\varepsilon_r = \ln(r/r_0)$ and natural thickness strain, $\varepsilon_t = \ln(t/t_0)$ are evaluated from the measurements taken with respect to the grid printed on the brass plate. Graphs of the principal natural hoop and thickness strain versus particle original radius are presented in Fig. 4.
(b) Photographs of some of the deformed profiles are shown in Fig. 3.

5. Finite element modelling procedure

Essentially, the modelling procedure addresses the issue of multiple contacts associated with rigid (die and blank holder) and deformable (plate and plug) parts. The general procedure adopted for the FE simulation consists of the following steps.

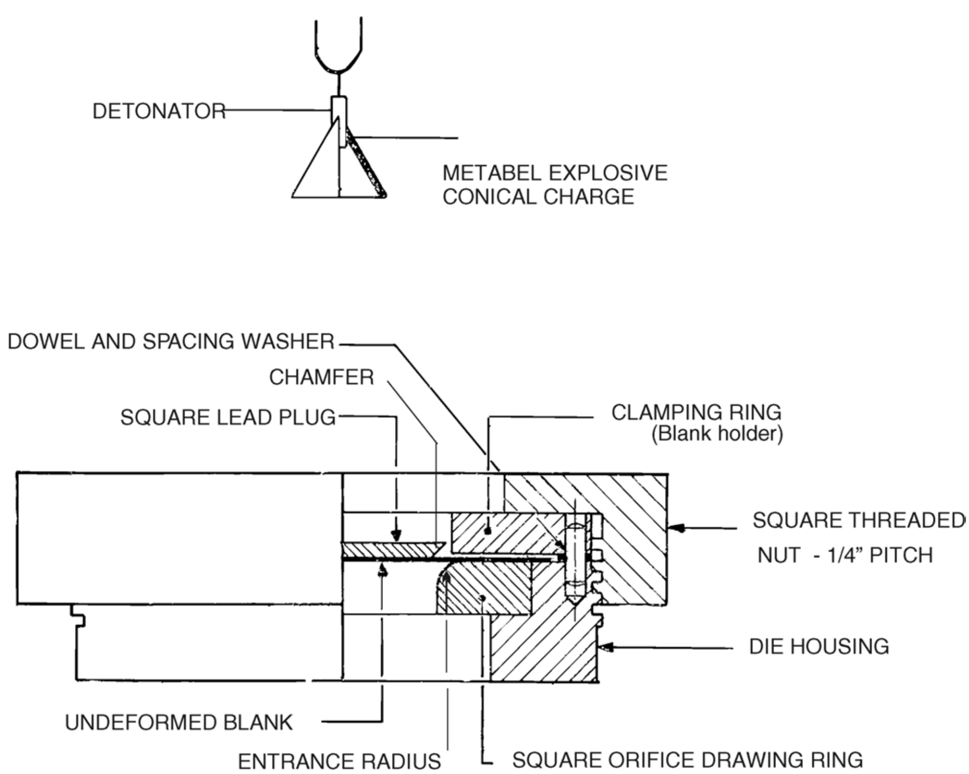

Fig. 2. Die details.