NUMERICAL INVESTIGATION OF SELF-PIERCING RIVETED DUAL LAYER JOINT

A Thesis by
UMA SHANKAR KRISHNAPPA

Bachelor of Engineering, UVCE, India, 2004

Submitted to the Department of Mechanical Engineering and the faculty of Graduate school of Wichita State University in partial fulfillment of the requirements for the degree of Master of Science

December 2008
DEDICATION

To my Family
I have examined the final copy of this thesis for form and content and recommend that it be accepted in partial fulfillment of the requirements for the degree of Master of Science, with a major in Mechanical Engineering

Hamid M. Lankarani, Committee Chair

We have read this thesis and recommend its acceptance:

Krishna Krishnan, Committee Member

Ramazan Asmatulu, Committee Member
DEDICATION

To my Family
ACKNOWLEDGEMENTS

I take this opportunity to express my sincere gratitude to my graduate advisor, Dr. HAMID M. LANKARANI, His constant encouragement and motivation helped me a lot in the completion of this thesis and my masters. I would also like to thank Dr. KRISHNA KRISHNAN and Dr. RAMAZAN ASMATULU for reviewing my thesis and making valuable suggestions.

I am indebted to my brother Manohar Krishnappa for his encouragement and suggestions throughout my career. I extend my gratitude to my parents, who supported me throughout my life and without their sacrifice and love I would not have achieved this goal.

Also, special thanks to Arun Gowda for his encouragement and suggestions throughout my thesis. I would like to acknowledge the support of my friends particularly Arunkarthik Soundararjan, Pradeep Ramaprasad, Anusha Garikapati and Sirisha Gokathoti.
ABSTRACT

Self-piercing riveting (SPR) is a high-speed mechanical fastening technique for point joining of sheet-material components. SPR is becoming important in automotive applications for aluminium vehicle body assembly. However, compared with current sheet-metal joining processes in the automotive industry, the effects of various parameters such as mechanical properties, rivet setting methods and systems, methods of removing self-piercing rivets, etc.

A study examining the effect of specimen configuration on the mechanical behavior of self-piercing riveted, dual-layer joints in aluminium alloys was conducted. It has observed that the specimen configuration had a significant effect on the strength and failure mechanism of a self-piercing riveted dual-layer joint.

The basic aspects of SPR process forming by conducting both explicit and implicit analysis have been investigated in this thesis. It was found that the operating force-deformation curve of SPR process was determined by the rivet deformation force and its displacement. Under certain process conditions, an increase in inertia effect due to high velocity of metal forming process results was not significant to an extent.

In this research, the springback characteristic parameters of the SPR process were also studied. The springback analysis carried out at the end of the forming process showed that the dimensional change in the part due to springback was not significant.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>CHAPTER</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Background</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Scope of the Research</td>
<td>4</td>
</tr>
<tr>
<td>2 LITERATURE REVIEW</td>
<td>5</td>
</tr>
<tr>
<td>2.1 Overview</td>
<td>5</td>
</tr>
<tr>
<td>2.1.1 Experimental test set-up</td>
<td>6</td>
</tr>
<tr>
<td>2.1.2 Numerical model</td>
<td>7</td>
</tr>
<tr>
<td>3 METHODOLOGY</td>
<td>9</td>
</tr>
<tr>
<td>3.1 Objective</td>
<td>9</td>
</tr>
<tr>
<td>3.2 Methodology</td>
<td>10</td>
</tr>
<tr>
<td>3.3 Proposed Numerical Investigation Model</td>
<td>12</td>
</tr>
<tr>
<td>4 FINITE ELEMENT MODELING OF SPR</td>
<td>14</td>
</tr>
<tr>
<td>4.1 Introduction to Finite Element Analysis</td>
<td>15</td>
</tr>
<tr>
<td>4.2 LS-DYNA Finite Element Code</td>
<td>15</td>
</tr>
<tr>
<td>4.3 FEA Simulation parameters</td>
<td>16</td>
</tr>
<tr>
<td>4.3.1 Description of model geometry</td>
<td>16</td>
</tr>
<tr>
<td>4.3.2 Simulation model geometry</td>
<td>18</td>
</tr>
<tr>
<td>4.3.3 Element formulation</td>
<td>21</td>
</tr>
<tr>
<td>4.3.4 Number of through integration points</td>
<td>21</td>
</tr>
<tr>
<td>4.3.5 Contact condition</td>
<td>22</td>
</tr>
<tr>
<td>4.3.6 Material modeling</td>
<td>22</td>
</tr>
<tr>
<td>4.3.7 Rigid body motion</td>
<td>24</td>
</tr>
<tr>
<td>4.3.8 Rigid body loading</td>
<td>24</td>
</tr>
<tr>
<td>4.3.9 Forming time</td>
<td>24</td>
</tr>
<tr>
<td>4.3.10 Adaptivity</td>
<td>25</td>
</tr>
<tr>
<td>4.3.11 Hourglass</td>
<td>26</td>
</tr>
<tr>
<td>5 SPRINGBACK ANALYSIS OF SPR</td>
<td>27</td>
</tr>
<tr>
<td>5.1 Introduction</td>
<td>27</td>
</tr>
<tr>
<td>5.2 Background</td>
<td>27</td>
</tr>
<tr>
<td>5.3 Methodology</td>
<td>29</td>
</tr>
<tr>
<td>5.4 Results</td>
<td>31</td>
</tr>
<tr>
<td>5.4.1 Induced springback force exerted</td>
<td>32</td>
</tr>
<tr>
<td>5.5 Summary</td>
<td>32</td>
</tr>
</tbody>
</table>
TABLE OF CONTENTS (Continued)

<table>
<thead>
<tr>
<th>CHAPTER</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 RESULTS AND DISCUSSIONS</td>
<td>33</td>
</tr>
<tr>
<td>6.1 Explicit Analysis</td>
<td>33</td>
</tr>
<tr>
<td>6.1.1 Comparison of force-deformation curve</td>
<td>33</td>
</tr>
<tr>
<td>6.1.2 Plotting of KE/IE ratio</td>
<td>34</td>
</tr>
<tr>
<td>6.1.3 Plotting of contact reaction force</td>
<td>34</td>
</tr>
<tr>
<td>6.1.4 Plotting of effective stress (V-M)</td>
<td>35</td>
</tr>
<tr>
<td>6.1.5 Plotting of effective strain</td>
<td>39</td>
</tr>
<tr>
<td>6.2 Implicit Analysis</td>
<td>43</td>
</tr>
<tr>
<td>6.2.1 Induced Von-Mises stress counters</td>
<td>43</td>
</tr>
<tr>
<td>6.2.2 Induced plastic strain counters at springback forming process</td>
<td>46</td>
</tr>
<tr>
<td>7 CONCLUSIONS AND RECOMMENDATIONS</td>
<td>50</td>
</tr>
<tr>
<td>7.1 Conclusions</td>
<td>50</td>
</tr>
<tr>
<td>7.2 Recommendations</td>
<td>51</td>
</tr>
<tr>
<td>7.2.1 Shearing Test</td>
<td>51</td>
</tr>
<tr>
<td>7.2.2 Tensile Test</td>
<td>51</td>
</tr>
<tr>
<td>REFERENCES</td>
<td>51</td>
</tr>
<tr>
<td>APPENDICES</td>
<td>54</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURE</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1</td>
<td>Self-piercing riveting process</td>
</tr>
<tr>
<td>Figure 1.2</td>
<td>A self-piercing rivet before installation</td>
</tr>
<tr>
<td>Figure 1.3</td>
<td>The backside of an installed SPR shows the impression left in the panel</td>
</tr>
<tr>
<td>Figure 2.1</td>
<td>Geometry of the testing device</td>
</tr>
<tr>
<td>Figure 2.2</td>
<td>Geometry of test device</td>
</tr>
<tr>
<td>Figure 2.3</td>
<td>Comparison between simulation and test (a) Numerical results; (b) Cross-section; (c) force-deformation curve</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>Sequential Methodology</td>
</tr>
<tr>
<td>Figure 3.2</td>
<td>Design and process modifications</td>
</tr>
<tr>
<td>Figure 4.1</td>
<td>Nomenclature of die and rivet geometry</td>
</tr>
<tr>
<td>Figure 4.2</td>
<td>Simulation Model Geometry</td>
</tr>
<tr>
<td>Figure 4.3</td>
<td>Simulation Model Geometry of Punch</td>
</tr>
<tr>
<td>Figure 4.4</td>
<td>Simulation Model Geometry of Pad</td>
</tr>
<tr>
<td>Figure 4.5</td>
<td>Simulation Model Geometry of Cylindrical Rivet</td>
</tr>
<tr>
<td>Figure 4.6</td>
<td>Simulation Model Geometry of Aluminium Alloy Sheet</td>
</tr>
<tr>
<td>Figure 4.7</td>
<td>Simulation Model Geometry of Die; DZ series</td>
</tr>
<tr>
<td>Figure 4.8</td>
<td>User defined integration rule and through the thickness integration Points</td>
</tr>
<tr>
<td>Figure 4.9</td>
<td>Punch load curve</td>
</tr>
<tr>
<td>Figure 5.1</td>
<td>Explicit-implicit sequential solution process</td>
</tr>
<tr>
<td>Figure 5.2</td>
<td>Resultant force due to springback process</td>
</tr>
<tr>
<td>Figure 6.1</td>
<td>Force-deformation curve of SPR rivet</td>
</tr>
</tbody>
</table>
# LIST OF TABLES

<table>
<thead>
<tr>
<th>TABLE</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 4.1 Definition of the specimens (Bolloff rivet and Die Identification)</td>
<td>18</td>
</tr>
<tr>
<td>Table 4.2 Rivet and Die parameters</td>
<td>18</td>
</tr>
<tr>
<td>Table 4.3 Appropriate Static and Dynamic Friction Coefficients</td>
<td>23</td>
</tr>
<tr>
<td>Table 4.4 Material parameters</td>
<td>24</td>
</tr>
<tr>
<td>Table 4.5 Mass Scaling</td>
<td>26</td>
</tr>
</tbody>
</table>
LIST OF SYMBOLS

Nomenclature

d  Diameter of rivet
D  Diameter of the die
E  Young’s module
F  Force applied to the punch
h  Die depth
L  Length of the rivet
t  Tip height

Greek letters

σ  effective stress
ε  effective strain
CHAPTER 1
INTRODUCTION

1.1 Background

Self-piercing riveting (SPR) is a relatively new joining method in the automotive field where it can be used to join thin sheet material [1, 10]. The technique involves clamping the sheets to be joined between a blank-holder and an upset die and forcing a rivet to pierce the upper sheet and flare into the bottom sheet under the influence of the upset die. The technique has many advantages. Unlike conventional riveting it does not require a pre-formed hole, thus saving on tooling, labour and time. As shown in Figure 1.1 the technique relies on forming a mechanical interlock rather than on fusion in can join a wide range of similar or dissimilar material combinations, where the use of resistance spot welding, for example, may be difficult or even impossible. Example of materials that can be joined by this technique include combinations of a wide range of steel and aluminium alloy grades, either uncoated or with a variety of surface treatments including pre-painted components.

![Figure 1.1: Self-piercing riveting process [1].](image)
The process can be applied to form multi-layer joints that may also include interlayer sealants or adhesives. In comparison to spot-welding, the process is environmentally friendly due to the low energy requirement, low-noise and absence of particulate and fume emissions, and it does not introduce heat into the components. Research in this area has shown that self-piercing riveting of aluminium alloys gives joints of comparable static strength and superior fatigue behavior to spot-welding. The use of lightweight material in the construction of automobile has found an increasing interest as a result of the feasibility of vehicle weight reduction and the improvement of fuel economy, drivability, and performance. Aluminium alloys offer considerable potential to reduce the weight of automobile body structures. The selection of an appropriate assembly technology for aluminium, however, is challenging in the design and manufacture of light-weight auto body structures. Self-piercing riveting (SPR) is one of the promising joining techniques for point joining of sheet-material components.

Figure 1.2: A self-piercing rivet before installation [9] Figure 1.3: The backside of an installed SPR shows the impression left in the panel [9]
The advantages of self-piercing riveting have offered the automotive industry the possibility of using lightweight alloys or multi-materials in body structure. The technique has allowed the development of new approaches in design that led to the manufacture of the Audi A8, the first generation of aluminium space frame structure [7], the Jaguar XJ type, and an all-aluminium monocoque body structure [8] and to the Volvo S80, a multi-materials application with integrated joining technologies [9]. In vehicle design, sheet material is the basic component that forms the body structure. These sheet materials are arranged by design engineers in various combinations in order to achieve the design engineers in various combinations in order to achieve the desired functions. The use of multi-layer arrangements is very common in body-in-white structures. Therefore, multi-layer joints with different sheet combination are very important from the design point of view. The process of self-piercing riveting components exhibited in Figure 1.2 and 1.3 has no particular problems in making multi-layer joints, usually up to a maximum of 3 thicknesses. Occasionally 4 thicknesses may be joined depending on the thicknesses of the individual sheets.

However, the mechanical behavior of multi-layer joints has not been examined by previous researches. In addition, the lack of a standard test method for self-piercing riveted joints makes it even more difficult to predict the real performance of multi-layer joints. It is therefore important to have a good understanding of the behaviors of multi-layer joints under different specimen configurations, in order to achieve the desired function of each joint. The aim of the study reported here was therefore to examine the effect of dual layer specimen configurations on the mechanical response and loading
characteristics of self-piercing riveted dual-layer joints. The information derived from this work may be useful to engineers in designing self-piercing riveted joints of optimum mechanical characteristics.

1.2 Scope of the research

In the thesis, study was carried on to present and discuss the possibility to simulate the riveting process using the commercial finite element code LS-Dyna. The simulations were performed using an explicit-implicit solution technique with r-adaptivity. Specimens made of aluminium alloy 6060 with T4 temper will be used for the validation of numerical simulations. Referring to the paper review sensitivity studies have been carried out in order to understand the influence of some important parameters such as friction, failure, mesh size, adaptive interval time, etc.

The FE model used in this analysis could be an alternative to the object-experiments. This technique could become a useful procedure for analyzing the stiffness and fatigue life of SPR specimen in the future. Tensile shear tests can be carried out in order to investigate the effect of work piece temperature and thickness on the static strength of SPR joints.
CHAPTER 2
LITERATURE REVIEW

2.1 Overview

In this paper [1], the riveting process has been simulated numerically using the finite element code LS-Dyna. The rivet and tools geometries are based on the Bollof standards. An implicit solution technique with r-adaptivity has been used. In addition, parametric studies on important parameters for the forming process, i.e. friction, mesh size and failure criteria are presented. In order to validate the numerical simulation of the riveting process, a new test device was developed in order to record the force applied on the rivet during the riveting process. An extensive experimental program on specimens made of aluminium alloy 6060 in two different tempers, T4 and T6, has been used as a database for the validation of the numerical simulations. The results also been shown the capability to simulate the riveting process for different combination of plate material and rivet geometries.

A thorough study of the self-pierce riveting process was done by Hahn and Dolle. An experimental test rig was established that enables force to be measured during the riveting process. Moreover, a numerical model of the riveting process was made using the finite element (FE) program MSC.AutoForge. Here, a geometric failure criterion on the minimum allowable sheet thickness was used to predict failure in the numerical model. In the paper [1] good agreement was found between the experimental force-deformation curves and the corresponding simulations.

Self pierce riveting of ultra high strength steel and aluminium alloy sheets is developed [3]. The deformation behavior in the riveting process is examined in finite element simulation and an experiment to evaluate optimum joining conditions.
2.1.1 Experimental test set-up

A new testing device was developed at SIMLab in order to investigate the riveting process, Figure. 2.1 and 2.2. The device is composed of the following parts: (1) punch, (2) blank holder, (3) die, (4) hydraulic system, (5) clamping bar. The die is the top part of a cylinder block made of high-strength steel that is screwed on the hydraulic piston, which is clamping bar. The black holder is mounted on a steel frame connected to the clamping bar.

![Figure 2.1: Geometry of the testing device [1]](image)

The tests were performed by fixing the test device into an Instron testing machine. The clamping bar is inserted and clamped into the machine. The specimen is positioned between the die toward the blank holder. The hydraulic system pushes the die toward the blank holder, clamping the specimen during the riveting process. Clamping pressure is
kept at a constant value of 80 bars during the process. The tests were performed under
displacement control. Mainly, this corresponds to a displacement of the punch equal to
the length of the rivet or a punch force of around 35-40KN.

Figure 2.2: Geometry of the test device [1]

Once the test has ended, the pressure between the blank holder and the die is
released and the specimen is extracted from the test device. Force-displacement histories
were recorded during the tests.

2.1.2 Numerical Model

The numerical model was validated against tests as shown in Figure 2.3. A visual
comparison between the numerical simulation and test is presented as well as a
comparison between the numerical and experimental force-deformation curves from the
riveting process. In the Figure 2.3, the borders of the numerical joint have been placed on
the picture of the cross-section of the specimen. The force-displacement curves have shown oscillations during the process, which is due to the high value of the penalty scale factor. A high value of the penalty scale factor is chosen in order to avoid contact problems.

![Image of cross-section and force-displacement curve]

Figure 2.3: Comparison between simulation and test (a) Numerical results; (b) Cross-section; (c) force-deformation curve. [1]

Good agreements between numerical and experimental results have been found for most of the specimens. In some cases a small divergence between the numerical and experimental tip of the rivet shank was observed. This may result in a difference in the last part of the force-displacement curve. Moreover, the gap between the rivet head and the top sheet is well described in the simulations for most of the specimens.
CHAPTER 3
METHODOLOGY

3.1 Objective

Self-Piercing Rivet (SPR) has been spotlighted in the automobile industry as an alternative method to resistance spot welding and arc welding, and the drawn designers’ keen interest as one of the potential lightening technology of the car body. These techniques are reliable in high volume production. SPR is a relatively quiet process that does not produce fumes and / or spatters, and requires less power than conventional techniques [13]. Thus, the SPR method may become more widely applied as one of the promising assembly methods for a wide range of materials such as coated, uncoated, pre-painted steels, combination of dissimilar sheets in automobile industry.

In this thesis, a case study of the SPR of dual sheets of aluminium Al6060 T4 temper alloy using a high strength steel rivet was simulated to numerical investigate. The model geometry is axisymmetric, which was developed and meshed using MSC. PATRAN. Numerical modeling of riveting is difficult, since it involves non-linearity such as damage, fracture and contact among deformable parts.

To generate and solve computational problems, finite element commercial coding solver LS-DYNA software was used. To validate force-deformation values were plotted comparing the obtained results from experimental laboratory test. Further to carry out the springback analysis at the end of the forming process showing the oscillatory change in shape of parts due to induced springback force was not significant.
3.2 Methodology

Points below depict the methodology from figure 3.1, carried out in this research:

- This study starts with the numerical investigation of a self-piercing riveting process using the finite element code LS-DYNA using an implicit time integrator.
- This explicit analysis problem is axisymmetric; the four-node 2D axisymmetric elements have been used.
- The high value of the penalty scale factor was chosen in order to avoid contact problems. Key measures were considered that with large value suppress springback deformation until very near to the termination time, making convergence during the first few steps easy.
- Also, care was taken to predict the small value of penalty scale factor may not stabilize the solution enough to allow equilibrium iteration to converge.
- As recommended R-adaptivity method in this study as the case of 2D axisymmetric was maintained. As the following reasons:
  - R-adaptivity preserves the logical structure of the grid
  - R-adaptivity remeshing is the only option for 2D axisymmetric
  - Completely new mesh will be initialized from the old mesh using least square approximation.
- In order to validate the riveting process, the force applied on the rivet during the numerical simulation of the riveting process was recorded and plotted cross wise with displacement of the rivet.
- Finally, implicit analysis was implemented to perform a springback analysis on the final configuration of the SPR joint right after explicitly formed FE model.
The springback implicit analysis of SPR was carried on to predict induced stress and strain contours and to simulate the release of the tooling force.

Figure 3.1: Sequential Methodology
3.3 Proposed Numerical Investigation Model

In general methodology for the study of the proposed cold forming technique included the following steps

- Modeling of finite element models using a pre-processor
  
  PATRAN and HYPERMESH were used as pre-processors depending on the complexity of the models to be generated. The developed geometries were then meshed with different options to generate the desired finite element models for carrying out of analysis.

- Analysis using a processor (Solver)
  
  LS-DYNA was used as the finite element analysis processor. LS-DYNA provides a wide variety of options to define various input parameters such as those required for contacts, material properties, etc.

  LS-DYNA being an explicit solver employs the direct integration technique, and also it is capable of providing accurate solutions for the dynamic equations of equilibrium that describe the behavior of the sheet in sheet metal forming problems.

- Post-processing the results to study the forming process
  
  LS-PREPOST was employed to post-process the results. The main features of the results that were of interest include

  * Plotting distribution of stress, strain and various parameters within the deformed sheet

  * Kinetic energy and internal energy ratios of deformed sheet metal.

  The ratio of kinetic energy to the internal energy of the sheet was used to ensure that the inertial effects in the analysis due to velocity and mass scaling are negligible. Other
several results like distribution of stresses and strain helped understand the response of the sheet in order to make modifications to improve the forming processes.

The first analysis trial of the riveting process was been simulated numerically using the finite element code LS-Dyna. In the Figure 3.2, the numerical investigation of self-piercing riveting processes with two blank sheets made of aluminium alloy 6060 with same tempers T4 were simulated.

In order to validate the numerical simulation of the riveting process, a new test processes was considered referring to an extensive experimental test results from the journal paper. Springback analysis was also carried out to study the change in the dimensional shape of the formed part due to elastic recovery.
4.1 Introduction to Finite Element Analysis

The finite element method can be described as a method of approximating complex functions by a summation of polynomials each valid within a small finite domain (element).

Basic steps of finite element analysis:

a. Modeling of a given structure by discrete finite elements. Finite elements can be either continuum elements that represent a volume of material or can be structural elements that mimic the behavior of basic structures such as rods, beams, shells, etc. In this step the number and type of finite elements that best approximate a given geometry are decided upon.

b. Selection of suitable polynomials (interpolation functions), for variables such as displacement, velocity, temperature, etc., to represent the variation of these quantities within the element.

c. Derive element characteristic matrices and load vectors. This step is not needed in commercial packages as these are pre-calculated. These characteristic matrices and load vectors describe the response of each type of element to loads and displacements imposed at the nodes.

d. Assemble the element matrices and vectors by matching the degrees of freedom (dof) into a global stiffness matrix (also mass and damping matrices for dynamic problems) and a global load vector.

\[
[K] \{X\} = \{F\}(M)\{x\} \quad \text{(4.1)}
\]
Again, the FEM package automatically does this step. Step 3 & 4 may be combined during manual formulation.

e. Apply boundary conditions to eliminate known dofs. Then solve the resulting equations to find out the displacements at all dofs.

f. Find element resultants (stresses, strains etc.) – These, and the displacements determined above, are the solutions of interest.

4.2 LS-DYNA Finite Element Code

The finite element code used for the purpose of analysis was generated from MSC PATRAN. LS-PREPOST which has options for editing the cards to prepare finite element code for processing in LS-DYNA. LS-DYNA is the processor and LS-PREPOST has the post processor.

Finite Element Analysis (FEA) involving simulating short-time large deformation dynamics, quasi-static problems with large deformations and multiple nonlinearities requires the use of either implicit or explicit solution techniques. LS-DYNA program includes the ability to address both implicit as well as explicit solutions. Explicit and implicit solution techniques or a combination of both have been used as the basis for many quasi-static finite element codes. Explicit computational algorithms typically used the central difference method for integration, wherein the internal and external forces are summed at each node point and a nodal acceleration is computed by dividing by the nodal mass. The solution is advanced by integrating this acceleration in time.
The major practical difference between the explicit and the implicit solution technique is the requirement on the time step size, $\Delta t$. The explicit solution technique is stable only if time-step $\Delta t$ is smaller than $\Delta t_{cr}$.

Therefore, $\Delta t_{cr}$ is the critical time step for shell elements, is given by

$$\Delta t_{cr} = \frac{Lc}{C} \quad (4.2)$$

Where $Lc$ is the characteristic length

$C$ is the speed of sound

The implicit method is not bound by the time-step size and is therefore unconditionally stable for larger time steps. Although the number of time steps required for an explicit solution can be larger than that of implicit methods. The major advantage of the explicit method is that there is no requirement for solving simultaneous equations, which means that no global matrix inversion is required, thus reducing computational time.

4.3 FEA Simulation parameters

4.3.1 Description of model geometry

The rivet is the Bollhoff type and is made of high strength steel with a nominal diameter of 5.0mm. This simple specimen geometry can be tested under different loading combinations in the test set-up presented by Porcaro et al.[1, 10]. The parameters selected were: (1) thickness of the top plate, (2) thickness of the bottom plate, (3) material of the top plate and finally (4) material of the bottom plate. For each combination of these parameters, the rivet geometry and the die shape were chosen in accordance with what is
used in the automotive industry today.

Figure 4.1: Nomenclature of die and rivet geometry [1]

A specimen’s type “sp1” with the combination of the different parameters is given in Table 4.2. Figure 4.1 shows the geometry of the rivet and die used in this investigation.

The Table 4.3 summarizes the rivet and die geometries for this test.

Table 4.1
Definition of the specimens (Bolloff rivet and die identification)

<table>
<thead>
<tr>
<th>Type</th>
<th>Thickness 1(mm)</th>
<th>Thickness 2(mm)</th>
<th>Material 1</th>
<th>Material 2</th>
<th>Die</th>
</tr>
</thead>
<tbody>
<tr>
<td>sp1</td>
<td>2</td>
<td>2</td>
<td>A6060 T4</td>
<td>A6060 T4</td>
<td>DZ 090 2025</td>
</tr>
</tbody>
</table>

Table 4.2
Rivet and die parameters

<table>
<thead>
<tr>
<th>TYPE</th>
<th>D (mm)</th>
<th>h (mm)</th>
<th>T (mm)</th>
<th>d (mm)</th>
<th>L (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>sp1</td>
<td>9.326</td>
<td>1.75</td>
<td>0.250</td>
<td>5</td>
<td>6</td>
</tr>
</tbody>
</table>
4.3.2 Simulation model geometry

Finite element geometry of complete axisymmetric global plane model is shown below in the Figure 4.2. A complete y-axisymmetric FE model of SPR process with x-y plane constrained globally.

\[ \text{Note: All dimension are in mm.} \]

Figure 4.2: Simulation Model Geometry
The pictorial view of FEA simulation model geometry parts are

I. **Punch**:

Finite element model geometry of punch with mesh size 0.5mm x 0.5 mm and dimensions of 5.337mm x 1mm. As shown in the Figure 4.3, a FE model of punch was generally meshed as it is being considered rigid body through the simulation.

![Figure 4.3: Simulation Model Geometry of Punch](image)  

II. **Pad**:

Finite element model geometry of pad with mesh size 0.8mm x 0.2 mm and dimensions of 4mm x 1mm. As shown in the Figure 4.4, a FE model of pad was developed considering as a rigid body.

![Figure 4.4: Simulation Model Geometry of Pad](image)  

III. **Sheet metal blank**:

Finite element model geometry of aluminium sheet with mesh size 0.1mm x 0.1 mm and dimensions of 10 mm x 2mm. As shown in the Figure 4.6, a single blank design with fine mesh size in order to understand the better sensitivity of mesh density.

![Figure 4.5: Simulation Model Geometry of Aluminium sheet](image)
IV. *Rivet*:

Finite element model geometry of cylindrical rivet with mesh size 0.1mm x 0.1 mm. As shown in the Figure 4.5, the mesh density was maintained in order to develop the cylindrical rivet profile according to Bolloff standard parametric values.

![Figure 4.6: Simulation Model Geometry of Cylindrical rivet](image)

V. *Die*:

Finite element model geometry of die was developed with paver mesh. One by bias method was adapted to mesh with varying mesh size. As shown in the Figure 4.7, an axisymmetric die profile was developed with fine mesh density. And as the FE model of Die was being considered rigid through analysis a reduced density of mesh was developed at the rest of die section.

![Figure 4.7: Simulation Model Geometry of Die; DZ series](image)
4.3.3 Element formulation

As the problem is 2D axisymmetric calculations, element formulation type EQ.14: Axisymmetric solid (Y-axis of symmetry)- area weighted was used. It is default shell element formulation in LS-DYNA. It is a computationally efficient element formulation used as common for all the axisymmetric geometric tools.

The type of element formulation for each part is specified in the *SECTION_SHELL card under ELFORM options.

4.3.4 Number of through integration points

To obtain better results the number of through thickness integration points, NIP value is specified using *SECTION_SHELL card. As the problem is axisymmetric, the default option four-node 2D axisymmetric elements have been used with four Gauss points. Fully integrated two-dimensional elements are available for options 13 and 15 by setting NIP equal to a value of 4 corresponding to a 2 by 2 Gaussian quadrature. Figure X.X shows the representation of the through thickness integration points.

Figure 4.8: User defined integration rule and through the thickness integration points [8]
4.3.5 **Contact condition:**

The *CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID* card was used to define contact between various parts, defined as master and slave. The part ID was used to identify the master (SIDM) and slave segments (SIDS). The part ID is specified in the *PART* card. The deformable body was considered as slave because it has to follow the shape of the master i.e. the rigid tools. Appropriate static and dynamic friction coefficients were specified as follows in the Table 4.3.

Table 4.3

<table>
<thead>
<tr>
<th>SIDS</th>
<th>SIDM</th>
<th>FS</th>
<th>FD</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>4</td>
<td>0.15</td>
<td>0.15</td>
</tr>
<tr>
<td>6</td>
<td>5</td>
<td>0.15</td>
<td>0.15</td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>0.3</td>
<td>0.3</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.3</td>
<td>0.3</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>0.15</td>
<td>0.15</td>
</tr>
</tbody>
</table>

4.3.6 **Material modeling**

In practice the tools are much stiffer than the blank; hence the tools can be treated as rigid bodies. The material model *MAT_020* in LS-DYNA was used to represent the rigid tools. The material properties for the tool are input in the *MAT_RIGID* card. The
boundary conditions for the rigid body were also specified in the same card. The following material properties were used for the rigid body:

- Mass density = 7.85E-9 ton/mm³
- Young’s modulus = 210 N/mm²
- Poisson’s ratio = 0.3

And the material model *MAT_024 in LS-DYNA was used to represent the rivet and sheet metals. The material properties for the tool are input in the *MAT_PIECEWISE_LINEAR_PLASTICITY card. The following Table 4.4 shows the material properties of rivet and sheet metals.

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s modulus E (Mpa)</th>
<th>Yield stress (Mpa)</th>
<th>Ultimate stress (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T4</td>
<td>69911</td>
<td>73</td>
<td>180</td>
</tr>
<tr>
<td>Rivet</td>
<td>188000</td>
<td>1520</td>
<td>1666.6</td>
</tr>
</tbody>
</table>

Plastic strain and corresponding yield stress values of aluminum alloys were specified in the sheet metal cards.
4.3.7 Rigid body motion

Displacement was given to the punch tool enforcing displacement on rivet using the *BOUNDARY_PRESCRIBED_MOTION_RIGID card. A cosine curve was specified for the motion of the tool as show in Figure 4.9. This was done to ensure that the punch tool does not impact the rivet towards the blank, which cause inertial effects resulting in erroneous results. The load curve was specified in the *DEFINE_CURVE card.

![Image of punch load curve]

Figure 4.9: Punch Load Curve

4.3.8 Forming time

Self piercing riveting is essentially a cold forming process carried out at low punch velocity. Simulating them at the same time scale becomes prohibitively costly computationally, hence for explicit finite element codes it is preferred to simulate the
process at higher velocities (velocity scaling). Alternatively the density of the material can also be increased to drive down the forming time (mass scaling) as shown in the Table 4.5.

However, care should be exercised to ensure that inertial effects do not affect the finite element solution. Therefore, mass scaling considering velocity of sound equation

$$U = \left(\frac{E}{\rho}\right)^{1/2} \quad (4.3)$$

Table 4.5
Mass scaling

<table>
<thead>
<tr>
<th>Type of Material</th>
<th>Actual Unit</th>
<th>Scaled with (10E+5) times</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density of Steel</td>
<td>7.85E-9</td>
<td>7.85E-4</td>
</tr>
<tr>
<td>Density of Aluminum</td>
<td>2.70E-9</td>
<td>2.70E-4</td>
</tr>
</tbody>
</table>

4.3.9 Rigid body loading

Desired loading force was applied on the aluminum sheets to stay clamped before riveting process starts. Loading force was defined using *LOAD_RIGID_BODY card. The load curve was specified in the *DEFINE_CURVE card with following chronological time steps.

4.3.10 Adapting

The adaptive process is used in LS-DYNA to obtain the greatest accuracy for a given set of computational resources. In adaptive method the elements are subdivided into smaller elements whenever an error indicator shows that subdivision of elements will
provide improved accuracy. The user sets the initial mesh size and the maximum level of 
adaptivity, and the program subdivides those elements in which the error indicator is the 
largest. The input parameters for the adaptivity process are specified in the 
*CONTROL_ADAPSTEP and *CONTROL_ADAPTIVITY CARDS. The part to be 
adaptively meshed is specified in the *PART card. In the simulations presented in this 
thesis, contact based adaptivity was used. In this type the mesh is adapted when the 
contact surface approach or penetrate the tooling surface depending on the value 
specified in the ADPENE option in the *CONTROL_ADAPSTEP CARD. Adaptivity is 
triggered by the change in the angle between adjacent elements.

For the 2D axisymmetric element, only the r-adaptivity method can be applied 
and thus used in this study. The adaptive options ADPOPT was given a value of 8 to 
specify 2D r-adaptive remeshing for axisymmetric and plan strain solid elements. When 
using this r-adaptivity in LS-Dyna, the user specifies a birth time, t birth, at which the 
adaptive remeshing begins, the death time, t death, at which the adaptive remeshing ends, 
and a time interval, Delta(t), between each remeshing. At each remeshing interval, nodal 
values for all variables to be remapped are generated. A completely new mesh is 
generated from the old mesh, based on the characteristic element size. The new mesh is 
initialized from the old mesh using a least squares approximation.

4.3.11 Hour glassing

Hourglass modes are a result of rank deficiency in the element stiffness matrix 
caused by insufficient integration points. These modes are not physically possible but can 
results in mathematical states. Hourglassing was controlled using Flanagan-Belytschko 
stiffness form by specifying IHQ=4 in the *HOURGLASS card.
CHAPTER 5
SPRINGBACK ANALYSIS

5.1 Introduction

In sheet metal cold forming process, springback deformation is an essential parameter that significantly complicates the design of improvement in die and process design achieving reduction in cost and improvement in the quality of the products. Springback can be best defined as the dimensional change of the formed part from that of the die, which occurs from elastic deformations during unloading. In most dynamic sheet metal forming operations, the highly non-linear deformation processes tend to generate a large amount of elastic strain energy in the blank material. This elastic strain energy, which becomes stored in the blank while it is in dynamic contact with the die, blank and piercing rivet component. It is subsequently released when the forming pressure is removed. This release of energy, which is the driving force for the springback, generally causes the blank to deform towards its original geometry. Therefore, the final part shape in sheet metal forming processes not only depends upon the contours of the die, but also on the amount of elastic energy stored in a part is a function of many parameters such as the material properties and the interfacial loads, predicting springback during forming is complicated.

Materials with high strength and high modulus of elasticity such as aluminium and high strength steel have high tendency for springback. Springback tends to increases with material strength and decreases with section modulus.

5.2 Background [8]

Researchers have been studying the phenomenon of springback for as long as four decades. A coupled explicit and implicit finite element method allows a designer to
efficiently characterize an entire sheet metal forming process to solve the forming and springback portions of an analysis. Narasimhan and Lovell [8] found the coupled finite element procedure could be utilized to significantly reduce the number of die prototypes designs that are currently required in sheet metal stamping operations. Hallquist et al. (1995) showed that the combination of the implicit solver for springback analysis with the explicit solver for the forming analysis provides a viable, accurate, and cost effective means of simulating even large sheet forming models. The rational for performing an explicit-to-implicit sequential solution is clear. Explicit methods are computationally efficient at solving the large plastic deformation and contact portions of the forming analysis. However springback calculations are very expensive with respect to processing time in the explicit regime. When the blank is removed from the tooling in the model, there is no way of increasing the speed of the analysis, in that there are no tools with which to force the blank to move more quickly. In order to remove oscillations which result, the explicit codes rely on dynamic relaxation, a process by which damping is applied to remove the kinetic energy. Selection of the correct damping constant is necessary for optimum convergence rates, Approaches to accelerate the dynamic relaxation process and hence to reduce the CPU times have been largely unsuccessful. Increasing the density will increase the time step, but the natural frequency of the sheet will be scaled by a similar amount, thus offsetting any potential decrease in the CPU time. Conversely, implicit codes are not well suited to solving interactions of a large number of nodes with rigid tooling, but they do handle the springback calculation very efficiently.
Shi et al. (1998) and Wagoner et al. (1996) found that in explicit dynamic forming simulations, the forming velocity influences springback prediction. Termination time was found to improve springback results by stabilizing the stress-state at the end of forming.

Shi et al. (1998) and Karafallis et al. (1996) found that the accuracy of bending and springback simulations are both increased when more through the thickness integration points are specified for shell elements. Springback simulations using seven to nine through the thickness integration points led to fairly accurate results.

5.3 Methodology

In this work the explicit and implicit codes have been coupled to accurately predict springback. Figure 5.1 schematically depicts the explicit-implicit sequential solution process that was utilized in this work.

The explicit finite element method was first used to solve the forming process. At the completion of the explicit analysis, the deformed shape, stresses and strains within the blank elements were transferred into the implicit portion of the program. This is done by including the *INTERFACE_SPRINGBACK_DYNA3D card in the LS-DYNA input deck for the forming simulation. This card allows LS-DYNA to write a file called dynain which contains the nodes and elements connectivity, the stress tensor, and the effective plastic strain for each integration point in the deformable material after the forming analysis. The dynain file serves as the input deck for the implicit stage of the analysis.

Implicit analysis in LS-DYNA can be accessed through the keywords which begin with the string *CONTROL_IMPLICIT. These may be included in the regular and
restart input decks. The restart capability allows modification of implicit control parameters in a springback simulation without repeating the forming simulation.

<table>
<thead>
<tr>
<th>Used LS-DYNA to explicitly solve the dynamic forming process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Used LS-DYNA to get result file outputs of containing stress, strain and displacement history of the blank elements</td>
</tr>
<tr>
<td>Applied boundary conditions are on the blank to restrain rigid body motion</td>
</tr>
<tr>
<td>Implicit solver initializes the stresses, strains and the displacements for the deformable blank</td>
</tr>
<tr>
<td>Implicit solver determines the deformation of the sheet during springback</td>
</tr>
</tbody>
</table>

Figure 5.1: Explicit-implicit sequential solution process
The *CONTROL_IMPLICIT_GENERAL keyword allows selection of the analysis type: explicit, implicit, or seamless explicit/implicit switching for springback. An implicit analysis can be run as either linear or non-linear using the keyword *CONTROL_IMPLICIT_NONLINEAR. Parameters for the sparse matrix linear equation solvers are entered in the *CONTROL_IMPLICIT_LINEAR keyword. For multi-step implicit simulations, the time or load step size may be either fixed or variable, according to the *CONTROL_IMPLICIT_AUTO keyword. A multi-step springback simulation can be performed using the artificial stabilization method to distribute unloading from springback over several steps. This feature is activated using parameters on the *CONTROL_IMPLICIT_STABALIZATION keyword.

One crucial consideration in building a static springback model is that enough support points (nodal constraints) must be included to support rigid body motion. In practice, these constraints define a reference point on the part, which remains fixed in space as the springback deformation occurs. All the springback deformations are then measured relative to this reference to this reference point.

5.4 RESULTS

The springback analysis was done to check for the dimensional change of the explicitly formed part after the release force of punch and the clamps holding the sheet.

The dynain file obtained at the end of the forming analysis was used as the input file. The springback analysis was used to predict the induced stress and strain values. Also, the springback force exerted to produce springback analysis.
5.4.1 Induced springback force exerted

Reaction force exerted from *DATABASE_RCFORCE card at explicit springback analysis as shown in the Figure 5.2. It is a reaction force plotted across time in order to simulate the release of the tooling force at SPR springback forming.

![Graph](image)

Figure 5.2: Resultant force due to springback process

5.5 Summary

The springback analysis was carried out by employing a coupled explicit-to-implicit sequential finite element method. The explicit solver was used for the forming analysis while the implicit solver was used for the springback analysis. The formed self-piercing riveting sheet metal forming was solved implicitly for the springback analysis and to save on computational time. Figure 6.6, shows stage by stage implicit springback forming process of induced V-mises stress counters. Where as Figure 6.7, shows the stage by stage implicit springback forming process of induced strain counters. The geometry of the part after springback superimposed compare to formed geometry showed.
CHAPTER 6
RESULTS AND DISCUSSION

6.1 Explicit Analysis

6.1.1 Comparison of force-deformation curve

The numerical simulated model was validated against the experimental test results as shown in Fig. 6.1. A visual comparison is carried on between the numerical simulation and test results of force-deformation curves from the riveting process. The force-deformation curves have shown oscillations during the process, which is due to high value of penalty scale factor. We can absorb that at displacement length after 1.5mm, the force has increased higher consistently than experimental result. The segment penetration between the mesh of rivet and top sheet was managed with high value of penalty scale factor than default value. Figure 6.4 and 6.5, shows the stage by stage SPR explicitly forming process of V-Mises stress and plastic strain respectively.

![Force-Deformation curve](image)

Figure 6.1: Force – deformation curve of SPR rivet
6.1.2 Plotting of KE/IE ratio

In the Figure 6.2, the plot shows the kinetic energy by internal energy across time. It has been absorbed that global KE/IE ratios is normalized after time 0.01s.

![Figure 6.2: Plotting of Inertial effect](image)

6.1.3 Plotting of contact reaction force

In the Figure 6.3, it shows that the contact reaction force exerting from punch and cylindrical rivet as master and slave respectively. *DATABASE_RCFORCE card was used to plot results.

![Figure 6.3: Plotting of Global Resultant Force](image)
6.1.4 Plotting of Effective Stress (V-M)

Figure 6.4: Effective Stress (V-M) of Explicit Analysis (Continued)
Figure 6.4: Effective Stress (V-M) of Explicit Analysis (Continued)
Figure 6.4: Effective Stress (V-M) of Explicit Analysis (Continued)
Figure 6.4: Effective Stress (V-M) of Explicit Analysis
6.1.5 Plotting of Effective Strain

Figure 6.5: Effective Strain of Explicit Analysis (Continued)
Figure 6.5: Effective Strain of Explicit Analysis (Continued)
Figure 6.5: Effective Strain of Explicit Analysis (Continued)
t = 0.192s

Figure 6.5: Effective Strain of Explicit Analysis
6.2 Implicit Analysis

6.2.1 Induced Von-Mises stress counters at springback forming process

Figure 6.6: V-M stress counters of Implicit Analysis (Continued)
Figure 6.6: V-M stress counters of Implicit Analysis (Continued)
Figure 6.6: V-M stress counters of Implicit Analysis
6.2.2 Induced plastic strain counters at springback forming process

Figure 6.7: Induced plastic strain of implicit analysis (Continued)
Figure 6.7: Induced plastic strain of implicit analysis (Continued)
Figure 6.7: Induced plastic strain of implicit analysis
CHAPTER 7
CONCLUSION AND RECOMMENDATIONS

7.1 Conclusions

Numerical investigation of self-piercing riveting (SPR) of dual layer joint was investigated with aluminium 6060 T4. The following conclusions were drawn from this thesis research.

In the SPR process, the force variation could be divided into two main stages: the rivet penetration region and the rivet-setting region. The penetrating force required to deform and penetrate the aluminium sheets was analyzed through explicit analysis. And the implicit analysis was carried out to predict the rivet-setting force, which is required to deform the rivet the form the final joint. Explicit approach is an efficient tool to study the joining process and predict the mechanical strength and failure mechanism of the SPR joints. Force-deformation values were predicted from rivet being riveted. And validation was analyzed with experimental results. Implicit approach is an efficient tool to study the SPR joining process and to predict the induced strains and residual stress right after forming of SPR in explicit approach. The relaxing forces predicted from implicit analysis were the force required to springback. For further studies the numerical investigation can be carried on with shear test and tensile test analysis of self-piercing riveting finite element model.

7.2 Recommendations

The research describes a numerical methodology (FE) for an efficient design of the self-piercing riveting process. Since the simulation give an overview of the whole process, problems can be found and new geometries or material combinations can be tested much easier then by experimental investigations.
7.2.1 Shearing Test [7]

The model of self-piercing riveting geometry is axisymmetric, but to perform the shear test a 3D model is necessary. Because of symmetry only half the specimen can be modeled.

![Fig 7.1: Isometric view of shear testing of self-piercing riveted joint [7]](#)

7.2.2 Tensile Test [7]

Tensile test with explicit approach is an efficient tool to study the joining process and predict the structural stiffness and fatigue life of the self-piercing rivet.

![Fig 7.2: Isometric view of tensile testing of self-piercing riveted joint [7]](#)
LIST OF REFERENCES


APPENDICES
APPENDIX A

Keyfile for Explicit analysis of Self-piercing rivet of a aluminium sheets

*KEYWORD

*TITLE

$# title

Thesis : Numerical Investigation of Self-Piercing Riveted Dual Layer Joints

*CONTROL_ADAPTIVE

$# adpfreq adptol adpopt maxlvl tbirth tdeath lcadp ioflag

1.0000E-8 1.00000 8 4 0.0001.0000E+20

$# adpsize adpass ireflg adpene adpth memory orient maxel

0.000 0 0 0.000 0.000 0 0 0

*CONTROL_BULK_VISCOSITY

$# q1 q2 type btype

1.500000 0.060000

*CONTROL_CONTACT

$# slsfac rwpnal islchk shlthk penopt thkchg orien enmass

1.000000 0.000 1 0 2 0 0

$# usrstr usrfrc nsbcs interm xpene ssthk ecdt tiedprj

0 0 0 0 4.000000

$# sfrie dfrie edc vfc th th_sf pen_sf

0.000 0.000 0.000 0.000 0.000 0.000 0.000

$# ignore frceng skiprwr outseg spotstp spotdel spothin

0 0 0 0 0 0.000

*CONTROL_CPU

$# cputim

0.000

*CONTROL_DYNAMIC_RELAXATION

$# nrcyck drtol drfctr drterm tssfdr irelal edttl idrflg

250 0.001000 0.9950001.0000E+30 0.900000 0 0.040000
*CONTROL_ENERGY
$# hgen  rwen  slnten  rylen
   1    2    1    1

*CONTROL_HOURGLASS
$# ihq  qh
   4    0.150000

*CONTROL_OUTPUT
$# npopt  neecho  nrefup  iaccop  opifs  ipnint  ikedit  iflush
   0    3
$# iprtf  ierode  tet10  msgmax  ipcurv
   0    0    0    0    0

*CONTROL_REMESHING
$# rmin  rmax  vf_loss  mfrac  dt_min
   0.100000  0.200000

*CONTROL_SHELL
$# wrpang  esort  irnxx  istupd  theory  bwc  miter  proj
  20.000000  2    -1    1    14    2    2
$# rotascl  intgrd  lamshl  cstyp6  tshell  nfail1  nfail4  psnfail
  1.000000    0    0    1

*CONTROL_TERMINATION
$# endtim  endcyc  dtmin  endeng  endmas
  0.200000

*CONTROL_TIMESTEP
$# dtinit  tssfac  isdo  tslimt  dt2ms  lctm  erode  ms1st
   0.000  0.6500000
$# dt2msf  dt2mslc  imslc
   0.000    0    0

*DATABASE_BNDOUT
$# dt  binary  lcur  ioopt
  2.0000E-4    1
*DATABASE_DEFGEO
$# dt binary lcur iopt
2.0000E-4 1

*DATABASE_DEFORC
$# dt binary lcur iopt
2.0000E-4 1

*DATABASE_GLSTAT
$# dt binary lcur iopt
2.0000E-4 3

*DATABASE_MATSUM
$# dt binary lcur iopt
2.0000E-4 3

*DATABASE_NCFORC
$# dt binary lcur iopt
2.0000E-4 1

*DATABASE_RCFORC
$# dt binary lcur iopt
2.0000E-4 3

*DATABASE_SECFORC
$# dt binary lcur iopt
2.0000E-4 1

*DATABASE_SPCFORC
$# dt binary lcur iopt
2.0000E-4 1

*DATABASE_EXTENT_BINARY
$# neiph neips maxint strflg sigflg epsflg rltflg engflg
0 0 0 0 1 1 1 1 1
$# cmpflg ieverp beamip dcomp shge stsz n3thdt ialemat
0 0 0 0 1 2 1

*BOUNDARY_PRESCRIBED_MOTION_RIGID
$# pid dof vad lcid sf vid death birth
*LOAD_RIGID_BODY

$# pid dof lcid sf cid m1 m2 m3
2 2 3 -40.000000

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID

$# cid title
1

$# sids sidm sfact freq fs fd dc membs
105 104 1.000000 50 0.150000 0.150000 0.000 6
$# tbirth tdeath sos som nds ndm cof init
0.0001.0000E+20 1.000000 1.000000 0 0 1

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID

$# cid title
2

$# sids sidm sfact freq fs fd dc membs
106 105 1.000000 50 0.100000 0.100000 0.000 6
$# tbirth tdeath sos som nds ndm cof init
0.0001.0000E+20 1.000000 1.000000 0 0 1

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID

$# cid title
3

$# sids sidm sfact freq fs fd dc membs
105 102 1.000000 50 0.300000 0.300000 0.000 6
$# tbirth tdeath sos som nds ndm cof init
0.0001.0000E+20 1.000000 1.000000

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID

$# cid title
4

$# sids sidm sfact freq fs fd dc membs
106 101 1.000000 50 0.300000 0.300000 0.000 6
$#
$t birth t death sos som nds ndm cof init
 0.000 1.000000 1.000000 1.000000

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID
$#
$#
cid
 5
$#
$sids sidm sfact freq fs fd dc membs
 104 103 1.000000 50 0.150000 0.150000 0.000 6
$#
$t birth t death sos som nds ndm cof init
 0.000 1.000000 1.000000 1.000000*PART
$#
$title
Die_prop
$#
$pid secid mid eosid hgid grav adpopt tmid
 1 1 1

*SECTION_SHELL
$#
$secid elform shrf nip propt qr/irid icomp setyp
 1 14 0.000 4 1 0 0 1
$#
$t1 t2 t3 t4 nloc marea idof edgset
 1.000000 1.000000 1.000000 1.000000

*MAT_RIGID_TITLE
Die_mat
$#
$mid ro e pr n couple m alias
 1 7.8000E-4 1.8800E+5 0.300000 0.000 0.000 0.000
$#
$cmo con1 con2
 1.000000 7 7
$#
$cmo or a1 a2 a3 v1 v2 v3
 0.000 0.000 0.000 0.000 0.000 0.000

*PART
$#
$title
Pad_prop
$#
$pid secid mid eosid hgid grav adpopt tmid
 2 2 3
*SECTION_SHELL

$#  secid  elform  shrf  nip  propt  qr/irid  icomp  setyp
  2  14  0.000  4  1  0  0  1

$#  t1  t2  t3  t4  nloc  marea  idof  edgset
  1.000000  1.000000  1.000000  1.000000

*MAT_RIGID>Title

Punch_pad_mat

$#  mid  ro  e  pr  n  couple  m  alias
  3  7.8000E-4  2.8800E+5  0.300000  0.000  0.000  0.000

$#  cmo  con1  con2
  1.000000  6  7

$#  lco  or  a1  a2  a3  v1  v2  v3
  0.000  0.000  0.000  0.000  0.000  0.000

*PART

$#  title

Punch_prop

$#  pid  secid  mid  eosid  hgid  grav  adpopt  tmid
  3  3  3

*SECTION_SHELL

$#  secid  elform  shrf  nip  propt  qr/irid  icomp  setyp
  3  14  0.000  4  1  0  0  1

$#  t1  t2  t3  t4  nloc  marea  idof  edgset
  1.000000  1.000000  1.000000  1.000000

*PART

$#  title

Rivet_prop

$#  pid  secid  mid  eosid  hgid  grav  adpopt  tmid
  4  4  4  0  0  0  2

*SECTION_SHELL

$#  secid  elform  shrf  nip  propt  qr/irid  icomp  setyp
  4  14  0.000  4  1  0  0  1
$#      t1        t2        t3        t4      nloc     marea      idof    edgset
1.000000  1.000000  1.000000  1.000000
*MAT_PIECEWISE_LINEAR_PLASTICITY_TITLE
Rivet_mat
$#     mid        ro         e        pr      sigy      etan      fail      tdel
4  7.7000E-4 1.8800E+5  0.300000 1520.0000 382.0000
$#       c         p      lcss      lcsr        vp
0.000     0.000         0         0     0.000
$#     eps1      eps2      eps3      eps4      eps5      eps6      eps7      eps8
0.000     0.000     0.000     0.000     0.000     0.000     0.000     0.000
$#     es1       es2       es3       es4       es5       es6       es7       es8
0.000     0.000     0.000     0.000     0.000     0.000     0.000     0.000
*PART
$#     title
Sheet1_prop
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
5         5         6         0         0         0         2
*SECTION_SHELL
$#     secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
5        14     0.000         4         1         0         0         1
$#      t1        t2        t3        t4      nloc     marea      idof    edgset
1.000000  1.000000  1.000000  1.000000
*MAT_PIECEWISE_LINEAR_PLASTICITY_TITLE
Sheet_mat
$#     mid        ro         e        pr      sigy      etan      fail      tdel
6  2.7000E-4 69910.000  0.350000 73.000000 135.0000
$#       c         p      lcss      lcsr        vp
0.000     0.000         0         0     0.000
*PART
$#     title
Sheet2_prop

$# pid secid mid eosid hgid grav adpopt tmid
   6   6   6   0   0   0   0   2

*SECTION_SHELL

$# secid elform shrf nip propt qr/irid icomp setyp
   6   14   0.000   4   1   0   0   0   1

$# t1 t2 t3 t4 nloc marea idof edgset
   1.000000 1.000000 1.000000 1.000000

*SET_NODE_LIST

$# sid da1 da2 da3 da4
   1

$# nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8
   310325 310360 310361 310362 310363 310364 310365 310366 310367 310368 310369

*SET_NODE_LIST

$# sid da1 da2 da3 da4
   2

$# nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8
   313279 313280 313281 313282 313283 313284 313285 313286 313287 313288 313289 313290 313291 313292 313293 313294 313295 313296 313297 313298 313299

*SET_NODE_LIST

$# sid da1 da2 da3 da4
   3

$# nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8
   313300 313401 313502 313603 313704 313805 313906 314007 314108 314209 314310 314411 314512 314613 314714 314815 314916 315017 315118 315219 315320

*SET_PART_LIST

$# sid da1 da2 da3 da4
    100
$# pid1 pid2 pid3 pid4 pid5 pid6 pid7 pid8
        4 5 6
*SET_SHELL_LIST_GENERATE
$# sid dal da2 da3 da4
        4
$# b1beg b1end b2beg b2end b3beg b3end b4beg b4end
        23530 25529
*CONSTRAINED_GLOBAL
$# tc rc dir x y z
        6 7 1
*INTERFACE_SPRINGBACK_LSDYNA
$# psid nshv dynain
        100
*END
APPENDIX B

Keyfile for Implicit analysis of Self-Piercing Rivet of dual aluminium sheets

*KEYWORD

*TITLE

$# title

Thesis : Numerical Investigation of Self-Piercing Riveted Dual Layer Joints

*CONTROL_ADAPTIVE

$# adpfreq adptol adpopt maxlvl tbirth tdeath lcadp ioflag

1.0000E-8  1.00000         8         4     0.0001.0000E+20

$# adpsize adpass ireflg adpene adpth memory orient maxel

0.000         0         0     0.000     0.000         0         0         0

*CONTROL_BULK_VISCOSITY

$# q1 q2 type

1.500000  0.060000

*CONTROL_CONTACT

$# slsfac rwpnal islchk shlthk penopt thkchg orien enmass

1.000000  0.000         1         0         2         0         0         0

$# usrstr usrfrc nsbcs interm xpene ssthk ecdt tiedprj

0         0         0         0  4.000000

$# sfrie dfrie edc vfc th th_sf pen_sf

0.000     0.000     0.000     0.000     0.000     0.000     0.000

$# ignore frceng skiprwg outseg spotstp spotdel spothin

0         0         0         0         0         0     0.000
*CONTROL_COUPLING

$#  unleng  untime  unforc  timidl  flipx  flipy  flipz subcyl
  1.000000  1.000000  1.000000  0.000  0  0  0  1

*CONTROL_CPU

$#  cputim

  0.000

*CONTROL_DYNAMIC_RELAXATION

$#  nrcyck  drtol  drfctr  drterm  tssfdr  irelal  edttl  idrflg
  250  0.001000  0.995000  1.0000E+30  0.900000  0  0.040000

*CONTROL_ENERGY

$#  hgen  rwen  slnten  ryle
  1  2  1  1

*CONTROL_HOURGLASS

$#  ihq  qh

  4  0.150000

*CONTROL_OUTPUT

$#  npopt  neecho  nrefup  iaccop  opifs  ipnint  ikedit  iflush
  0  3

$#  iprte

  0

*CONTROL_REMESHING

$#  rmin  rmax

  0.100000  0.300000
*CONTROL_SHELL

$#  wrpang esort irnxx istupd theory bwc miter proj

20.000000  2  -1   1   14   2   2

*CONTROL_TERMINATION

$#  endtim endcyc dtmin endeng endmas

0.100000

*CONTROL_TIMESTEP

$#  dtinit tssfac isdo tslimt dt2ms lctm erode ms1st

0.000  0.900000

$#  dt2msf dt2mslc imsc

0.000  0  0

*DATABASE_BNDOUT

$#  dt  binary

0.001000  1

*DATABASE_DEFGEO

$#  dt  binary

0.001000  1

*DATABASE_DEFORC

$#  dt  binary

0.001000  1

*DATABASE_GLSTAT

$#  dt  binary

0.001000  3
*DATABASE_MATSUM
$# dt binary
 0.001000 3

*DATABASE_NCFORC
$# dt binary
 0.001000 1

*DATABASE_RCFORC
$# dt binary
 0.001000 3

*DATABASE_SECFORC
$# dt binary
 0.001000 1

*DATABASE_SPCFORC
$# dt binary
 0.001000 1

*DATABASE_BINARY_D3PLOT
$# lcut beam npltc
 0.001000

$# ioopt
 0

*DATABASE_EXTENT_BINARY
$# neiph neips maxint strflg sigflg epsflg rltflg engflg
 0 0 0 1 1 1 1 1 1
<table>
<thead>
<tr>
<th>cmpflg</th>
<th>ieverp</th>
<th>beamip</th>
<th>dcomp</th>
<th>shge</th>
<th>stssz</th>
<th>n3thdt</th>
<th>ialemat</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

*BOUNDARY_PRESCRIBED_MOTION_RIGID*

<table>
<thead>
<tr>
<th>pid</th>
<th>dof</th>
<th>vad</th>
<th>lcid</th>
<th>sf</th>
<th>vid</th>
<th>death</th>
<th>birth</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>2</td>
<td>2</td>
<td>1</td>
<td>-7.250000</td>
<td>01.0000E+10</td>
<td>0.023000</td>
<td></td>
</tr>
</tbody>
</table>

*LOAD_RIGID_BODY*

<table>
<thead>
<tr>
<th>pid</th>
<th>dof</th>
<th>lcid</th>
<th>sf</th>
<th>cid</th>
<th>m1</th>
<th>m2</th>
<th>m3</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>2</td>
<td>3</td>
<td>-4.000000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID*

<table>
<thead>
<tr>
<th>cid</th>
<th>title</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>sids</th>
<th>sidm</th>
<th>sfact</th>
<th>freq</th>
<th>fs</th>
<th>fd</th>
<th>dc</th>
<th>membs</th>
</tr>
</thead>
<tbody>
<tr>
<td>105</td>
<td>104</td>
<td>1.000000</td>
<td>50</td>
<td>0.150000</td>
<td>0.150000</td>
<td>0.000</td>
<td>6</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>tbirth</th>
<th>tdeath</th>
<th>sos</th>
<th>som</th>
<th>nds</th>
<th>ndm</th>
<th>ipf/cof</th>
<th>init</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0001</td>
<td>1.0000E+20</td>
<td>1.000000</td>
<td>1.000000</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID*

<table>
<thead>
<tr>
<th>cid</th>
<th>title</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>sids</th>
<th>sidm</th>
<th>sfact</th>
<th>freq</th>
<th>fs</th>
<th>fd</th>
<th>dc</th>
<th>membs</th>
</tr>
</thead>
<tbody>
<tr>
<td>106</td>
<td>105</td>
<td>1.000000</td>
<td>50</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.000</td>
<td>6</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>tbirth</th>
<th>tdeath</th>
<th>sos</th>
<th>som</th>
<th>nds</th>
<th>ndm</th>
<th>ipf/cof</th>
<th>init</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0001</td>
<td>1.0000E+20</td>
<td>1.000000</td>
<td>1.000000</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

68
*SET_PART_LIST

$# sid da1 da2 da3 da4
106

$# pid1 pid2 pid3 pid4 pid5 pid6 pid7 pid8
6

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID

$# cid title
3

$# sids sidm sfact freq fs fd dc membs
105 102 1.000000 50 0.300000 0.300000 0.000 6

$# tbirth tdeath sos som nds ndm ipf/cof init
0.0001.0000E+20 1.000000 1.000000

*SET_PART_LIST

$# sid da1 da2 da3 da4
102

$# pid1 pid2 pid3 pid4 pid5 pid6 pid7 pid8
2

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID

$# cid title
4

$# sids sidm sfact freq fs fd dc membs
106 101 1.000000 50 0.300000 0.300000 0.000 6

$# tbirth tdeath sos som nds ndm ipf/cof init
*SET_PART_LIST

$# sid  da1  da2  da3  da4
   101
$# pid1  pid2  pid3  pid4  pid5  pid6  pid7  pid8
   1

*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_ID

$# cid title
   5
$# sids  sidm  sfact  freq  fs  fd  dc  membs
   104  103  1.000000  50  0.150000  0.150000  0.000  6
$# tbirth  tdeath  sos  som  nds  ndm  ipf/cof  init
   0.000  1.000000  1.000000  1.000000

*SET_PART_LIST

$# sid  da1  da2  da3  da4
   103
$# pid1  pid2  pid3  pid4  pid5  pid6  pid7  pid8
   3

*PART

$# title
Die_prop

$# pid  secid  mid  eosid  hgid  grav  adpopt  tmid
   1  1  1
*SECTION_SHELL

$# secid elform shrf nip propt qr/irid icomp setyp
  1  14  0.000  4  1  0  0  1
$# t1 t2 t3 t4 nloc marea
  1.000000  1.000000  1.000000  1.000000

*MAT_RIGID_TITLE

Die_mat

$# mid ro e pr n couple m alias
  1  7.8000E-4 1.8800E+5  0.300000  0.000  0.000  0.000
$# cmo con1 con2
  1.000000  7  7
$# lco or a1 a2 a3 v1 v2 v3
  0.000  0.000  0.000  0.000  0.000  0.000

*PART

$# title

Pad_prop

$# pid secid mid eosid hgid grav adpopt tmid
  2  2  3

*SECTION_SHELL

$# secid elform shrf nip propt qr/irid icomp setyp
  2  14  0.000  4  1  0  0  1
$# t1 t2 t3 t4 nloc marea
  1.000000  1.000000  1.000000  1.000000
*MAT_RIGID_TITLE

Punch_pad_mat

$#     mid        ro         e        pr         n    couple         m    alias
3 7.8000E-4 2.8800E+5  0.300000     0.000     0.000     0.000
$#     cmo      con1      con2
1.000000  6  7
$#lco or a1       a2        a3        v1        v2        v3
0.000     0.000     0.000     0.000     0.000     0.000

*PART

$# title

Punch_prop

$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
3         3         3

*SECTION_SHELL

$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
3        14     0.000         4         1         0         0         1
$#      t1        t2        t3        t4      nloc     marea
1.000000  1.000000  1.000000  1.000000

*PART

$# title

Rivet_prop

$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
4         4         4         0         0         0         2
*SECTION_SHELL

$# secid elform shrf nip propt qr/irid icomp setyp
4 14 0.000 4 1 0 0 1

$# t1 t2 t3 t4 nloc marea
1.000000 1.000000 1.000000 1.000000

*MAT_PIECEWISE_LINEAR_PLASTICITY

Rivet_mat

$# mid ro e pr sigy etan fail tdel
4 7.7000E-4 1.8800E+5 0.300000 2520.0000 382.00000

$# c p lcse lcsr vp
0.000 0.000 0 0 0.000

$# eps1 eps2 eps3 eps4 eps5 eps6 eps7 eps8
0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000

$# es1 es2 es3 es4 es5 es6 es7 es8
0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000

*PART

$# title

Sheet1_prop

$# pid secid mid eosid hgid grav adpopt tmid
5 5 6 0 0 0 2

*SECTION_SHELL

$# secid elform shrf nip propt qr/irid icomp setyp
5 14 0.000 4 1 0 0 1

73
$# t1 t2 t3 t4 nloc marea
1.000000 1.000000 1.000000 1.000000

*MAT_PIECEWISE_LINEAR_PLASTICITY_TITLE

Sheet_mat

$# mid ro e pr sigy etan fail tdel
6 2.7000E-4 69910.000 0.350000 73.000000 135.000000

$# c p lcse lcsr vp
0.000 0.000 0 0 0.000

*PART

$# title

Sheet2_prop

$# pid secid mid eosid hgid grav adpopt tmid
6 6 6 0 0 0 2

*SECTION_SHELL

$# secid elform shrf nip propt qr/irid icomp setyp
6 14 0.000 4 1 0 0 1

$# t1 t2 t3 t4 nloc marea
1.000000 1.000000 1.000000 1.000000

*SET_NODE_LIST

$# sid da1 da2 da3 da4
1

$# nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8
310325 310360 310361 310362 310363 310364 310365 310366
*SET_NODE_LIST

$#   sid   da1   da2   da3   da4

   2

$#   nid1   nid2   nid3   nid4   nid5   nid6   nid7   nid8
   313279  313280  313281  313282  313283  313284  313285  313286
   313287  313288  313289  313290  313291  313292  313293  313294
   313295  313296  313297  313298  313299

*SET_NODE_LIST

$#   sid   da1   da2   da3   da4

   3

$#   nid1   nid2   nid3   nid4   nid5   nid6   nid7   nid8
   313300  313401  313502  313603  313704  313805  313906  314007
   314108  314209  314310  314411  314512  314613  314714  314815
   314916  315017  315118  315219  315320

*SET_PART_LIST

$#   sid   da1   da2   da3   da4

   100

$#   pid1   pid2   pid3   pid4   pid5   pid6   pid7   pid8
   4     5     6

*SET_SHELL_LIST_GENERATE

$#   sid   da1   da2   da3   da4

   4
$# b1beg  b1end  b2beg  b2end  b3beg  b3end  b4beg  b4end

23530  25529

*CONSTRAINED_GLOBAL

$#  tc  rc  dir  x  y  z

6    7   1

*DAMPING_GLOBAL

$#  lcid  valdmp  stx  sty  stz  srx  sry  srz

0  0.000  0.000  0.000  0.000  0.000  0.000  0.000

*INTERFACE_SPRINGBACK_LSDYNA

$#  psid  nshv  dynain

100

*END