Journal of Engineering Research and Applied Science
Available at www.journaleras.com
Volume 2(2) December 2013, pp 144-153
ISSN 2147-3471 © 2013

A novel design to minimize errors due to trimming in sheet metal forming software

Oguz Kayabasi[1,a]

[1]Duzce University, Biomedical Engineering, Duzce, Turkey.

Accepted 20 Sep 2013

Abstract
The forming operations start with forming of metal and then trimming the unnecessary part and finally spring back calculations. After completing forming operation, by means of a curve defining trimming boundaries, unnecessary parts are trimmed, similar to the real operations. However in numerical environment after trimming new mesh generation around the trimming boundary is unavoidable. The strains calculated for forming operation are mapped on to the new mesh generated after trimming. The new nodes created can not find any corresponding strains. Although the number of new node, are not too much with respect to the total number of nodes on a sheet formed, they are the main causes of errors in spring back operations. In this study to overcome this problem a new approach is recommended. In this approach., before trimming operation, the strains found on the sheet after forming are used to find SR (surface-regression) for the studied sheet. Since SR can define the strains for any x, y and z coordinate, the nodes after trimming operations can find approximated strain values. This method was applied on real parts and it was found that the errors are almost disappeared.

Keywords: Trimming; spring back; surface regression

1. Introduction

In the sheet metal forming operations, due to the nature of operation or depending on the design of product or material behaviour, the process must be divided into stages, such as in multistage forming. But most of this operations need to calculate spring back after forming. However the accuracy of calculation depends on data used. It was seen that in large strain dynamic analyses codes such as Ls-Dyna 970, the errors coming from new nodes created during trimming operation are evident and cited in their manuals [1].

The spring back calculations may be done either as a part of forming calculations or after forming operations [2-6]. In explicit codes the forming operations was performed explicitly. At the end of analysis a file was created. This file contains node, element and stress information for forming operations. The second step was trimming operation. In this operation the previous file is used together with a trimming curve definition. This curve definition is used to trim the part surfaces. There are many researches for trimming surfaces.

The use of combined subdivision schemes is guaranteed exact interpolation of trim curves for sheet forming operations [7]. Although exact interpolation of trimming curves is obtained, element distortions due to trimming must be handled to give proper topological information. A vertex based tolerant approach was proposed to obtain correct topology [8,9]. Trimmed surfaces must be remeshed. At least the trimmed regions must be modified. Automatic remeshing methods are generated [10]. But the remeshing is not suitable for progressive forming operations because of the necessity of mapping of data between operations. The data of previous operation must be used for the next operation. The mapping algorithms are generated [11]. Trimmed surface cad models were employed for automatic triangulation [12].

The importance of mapping comes from the fact that inappropriate integration of stresses may give rise to numerical instabilities. Especially for impact simulations or high velocity deformations the mapping becomes important [13].

[1,a] Corresponding author;
Phone: +90-380-542-1036, Email : oguzkayabasi@duzce.edu.tr

There are many remeshing techniques regarding desired modifications on the mesh. For the mapping of data, lower bound limit mapping algorithm is generated. The procedure is based on a parametric mapping technique, coupled with midpoint splitting of subdomains, and permits the user to control the distribution of the discontinuities and elements precisely. Although it is not fully automatic, the algorithm is fast and automatically generates extension zones for problems with semi-infinite domains [14]. Automatic mesh generation with desired user input is also controlled by means of intelligent local approach [15].

The remeshing techniques play important role on sheet forming operations. Finite-element analysis of large-strain problems involves severe mesh distortion in the course of modelling the material deformation. After some stage of deformation, the initial mesh becomes totally unacceptable from the accuracy point of view. At this stage, a proper remeshing and accurate transfer of state parameters to the new mesh has to be carried out to ensure proper process simulation. Updated lagrangian formulation is considered in some work [16]. Two basic methods of transferring the smoothed old nodal data to the Gauss points of the new mesh are studied. The problem of violating the consistency condition that is associated with data transfer from the old mesh to the new mesh, and the effects of such violation, are discussed.

The second analysis is similar to the forming analysis but there is no force and time for process because of this operation is for only trimming. The trimming operation also creates a file similar to the first one. But in this file there are new nodes and elements created around the trimming boundary. The stress values of these nodes are zero. At the final step the new file is used for implicit spring back calculations [17,18].. This discrete method was used in the early version of programs such as LS-Nike.3D. However in new versions the forming, trimming and spring back calculations can be done at one analysis. This new versions can able to use explicit and implicit solution techniques in the same analyses.

Although the finite element codes were improved, errors coming from new nodes still play important role in the calculation.. In the study, the method applied needs some calculations similar to the steps described above. This approach has five steps, first of all forming is performed explicitly, and then SR method is used to find the relation between coordinates and stress or strains. When the necessary equations are found, the third step which is trimming performed. Then the stress and strain values are found for the new nodes generated during forming and written to file. Finally spring back calculations are performed without any errors. The method was implemented to the calculations performed by means of Ls-Dyna 970 code.

2. Material and Methods

Many different parts were used in study. In order to eliminate other factors on errors encountered during spring back calculations, same material model (piecewise linear plasticity) and same material (Al 2012) alloy were used.

Forming operation for a part is done by using die and punch surfaces written in VDA or IGES files. Shape of blank is flat. The contacts between blank-die and blank-punch are created by means of suitable contact algorithms. One-way forming contact algorithms are used in general. Then velocity as a function of time is defined by a load curve and applied to punch. At the end of analyses, the new coordinates of nodes and elements together with stress and strains are written to a file. Finally this file is used for spring back calculations.

Spring back calculations are done according to procedure described above. However, in practice, the spring back of the final part needs to be modelled for the condition where the flanges have been removed. For this reason, a means of removing excess material from the blank must be determined before the calculation of spring back response. Removing the extra material can seriously effect the spring back of the part.

The removing of excess material is done by means of trimming algorithms. It is important that these trimming algorithms do not simulate an actual trimming operation in the sense that there is no blanking die contacting the sheet.

As a numerical method, trimming is not caused by material fail instead a “fast and dirty” approach is used in which elements which lie within the region to be trimmed are simply deleted. Nodes, which no longer belong to an element, are also deleted.

To explain the trimming operation as a numerical mean following figure 1 is shown. Consider a mesh of elements, which has an internal area to be trimmed out. Some elements lie within the area to be deleted, some are outside, and some are on the border.

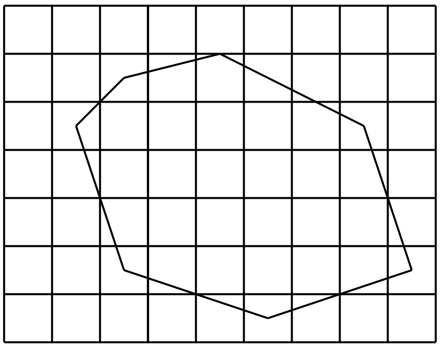

Figure 1. Mesh considered for trimming

Those elements which lie entirely inside the trim curves could only be chosen to delete as shown in figure 2. Another choice is that deleting elements which lie in or on the boundary. These elements are shown in figure 3. Final choice for element deletion is to select the elements whose centroids lie within the trim curves. These elements are shown in figure 4 also. It is possible to delete elements whose centroids lie outside the trim curve. However, whatever choice is selected, the problem that the mesh, which results is pretty ugly, still exists.

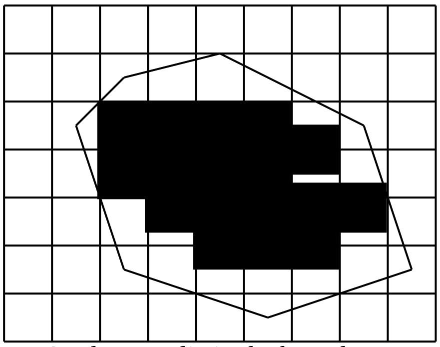

Figure 2. Elements lie in the boundary.

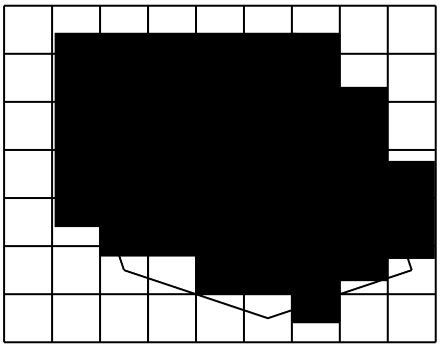

Figure 3. Elements lie in or on the boundary

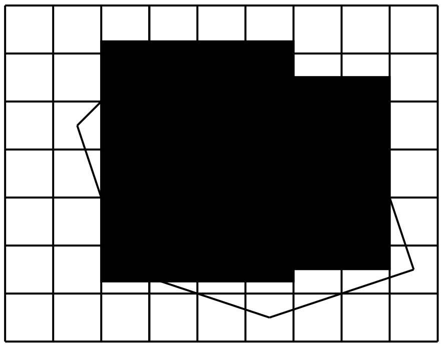

Figure 4. Elements whose centroids lie within the trim curves

The mesh, which is created by simply deleting elements is not only lacking in aesthetic appeal, it also is not a good mesh for predicting spring back. Nodes, which are connected to only one element, can have unrealistically large spring back because they are quite unrestrained.

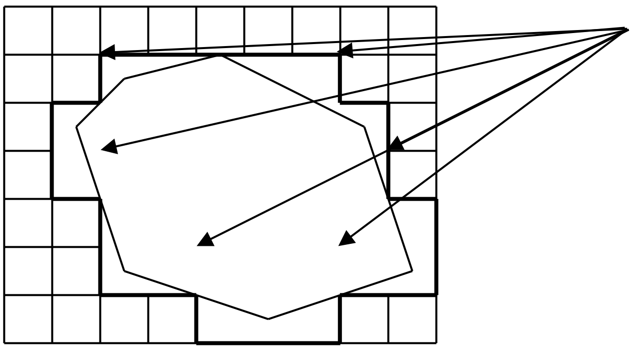

Figure 5. Elements deleted.

These nodes are shown in figure 5. Instead it would be better to delete elements, which entirely lie inside the trim curve, and modify elements, which the curve passes through. Modification means splitting element by means of trim curve. This operation is shown in figure 6.

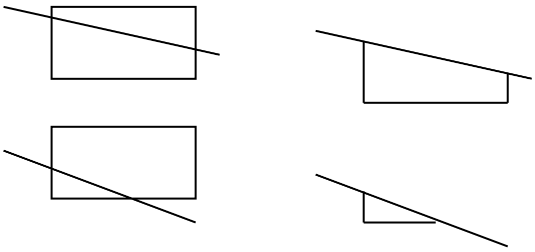

Figure 6. Trimming of elements

Ideally, a good solution for the msh trimming problem should satisfy the following requirements;
Correctness; The computed trimming curves must lie on the surface. If the trimming curves lie only on the faces of the meshes but not on the surfaces, then the finite element analysis will yield false or unacceptable results. The easiest way to guarantee correctness is to compare the trimming curves in

parametric space and then map them to object space. Faithfulness; The geometry of the resulting surface trust faithfully reflect the geometry of the original surface, because they represent the intent of the designer. In particular, new parametric paches should not be defined using the trimming curves, because this would yield a different geometry.

3. The algorithm

The algorithm used in this work for trimming surface has two basic steps, which aim at fulfilling the requirements.

a. Determination of the trimming curves
    a. Link intersecting points into polygonal lines representing the trimming curves
    b. Interpolate parametric curve passing through polygonal line points
    c. Compute new points with proper spacing on the curves and
    d. Move those new points onto surface

b. Topology reconstruction
    a. Determine the trimming regions removing vertices and edges based on polygonal lines
    b. Insert new edges over the trimming curves using the new points defined above
    c. Triangulate the trimming regions on surface
    d. Smooth mesh

In step (a) trimming curves are defined or computed. If the trimming curves are the intersection of two different surface then the above algorithm becomes 3 step algorithm . This time at first the intersecting lines must be computed. However, in general for trimming operations the trimming curve is defined by designer at the beginning of trimming operation. This curve is defined as point in usual. Therefore the algorithm reads these data at the first step. Then convert them in parametric space Equally spaces sample points are preferred.

In step (b) trimming regions on surface are identified. These regions, actually sub-patches, are faces of the topological data structure generated by the elimination of some edges. At the end of this step on surface there are as many regions as the number of trimming curves. The insertion of edges representing trimming curves leads the subdivision of trimming surface in two different sub-surfaces. Each trimmed region is triangulated by, inserting the edges.

Finally, to increase the shape quality of the faces generated the standard laplacian smoothing technique in parametric space is used. Therefore, the coordinates of each parametric vertices is changed at an average of coordinates of its neighbour. The averaging is repeated a number of times. This is the usual way of smoothing [19].

Previous trimming algorithms implemented to old version commercial software can lead to some rather ugly looking individual elements. Currently, some intelligence is added which is intended to combine triangles with adjacent quads to create a larger quad. Also, small quads will be added to adjacent larger quads. However, there are some constraints for these operations, such as, elements must not be inverted and should not be so small as to severely limit the time step. The new mesh created after trimming is shown in figure 7.

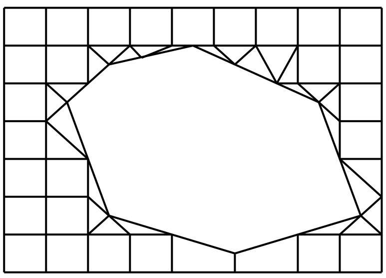

Figure 7. New mesh after trimming.

After successfully forming a part, it must be trimmed. This may be a pre-cursor to a spring back operation, or it may be necessary for a crash analyses. The trimming capability of current large strain dynamic analyses explicit codes is not about simulating metal fracture. Rather, elements outside the final part are simply deleted from the input file. Elements on the trim boundary are split. Occasionally, some new elements are created. Newly created elements will be given zero stress and strain, which can lead to problems in spring back analysis. The lack of stress and strain for newly created nodes after trimming operation is the main focus of this study. The errors during spring back calculations were encountered discouraged the use of large deformation dynamic analyses explicit codes for simulating forming operation when the spring back is important. Since the cost of die manufacturing is too high to accept high risk, the accuracy of spring back calculations becomes more and more important.

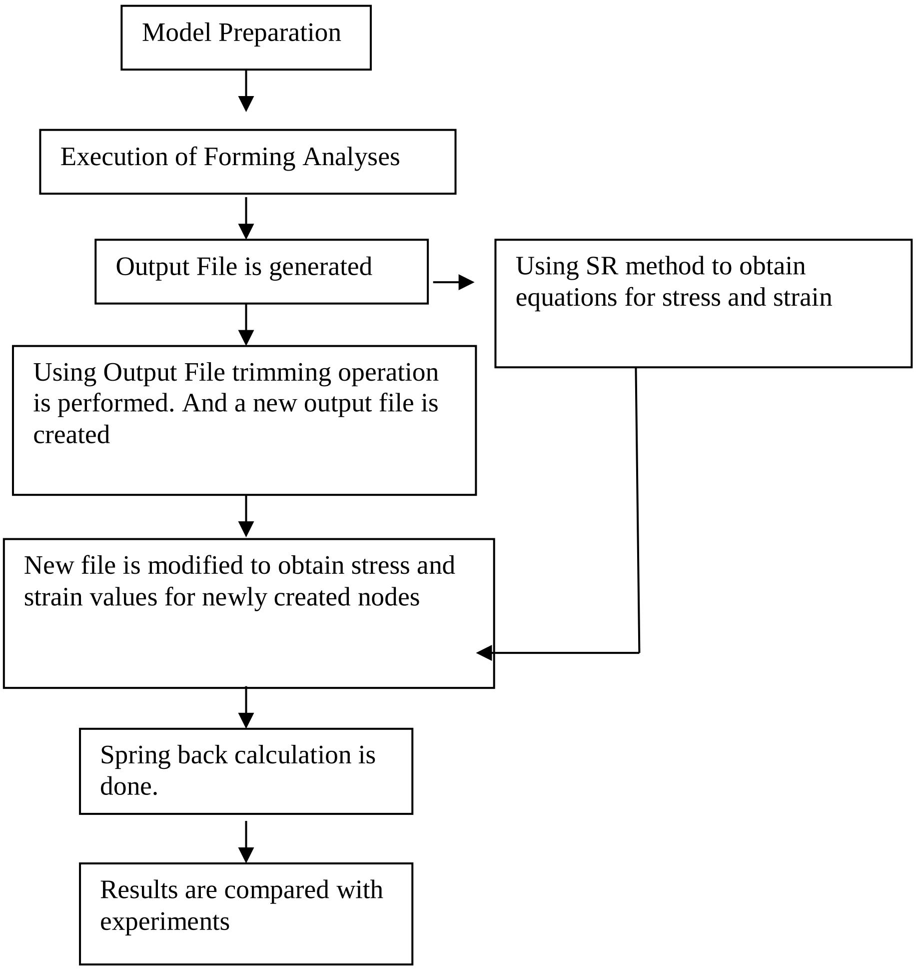

Figure 8. Flow chart of operation sequences

In the study the stress and strain values obtained after forming operation are written to a file. This file is used to obtain stress and strain surfaces by means of SR method. The surface regression of these data provides the equations defining stress or strain values as a function of x,y and z coordinates. Then the file is used to obtain trimming operation as usual. The third step is to find the stress and strain values for newly created nodes. Finally spring back calculation is completed. This procedure is given in figure 8 as a flow chart. This approach is used on forming of many automotie panel parts. One of them is given in figure 9.

Figure 9. Front panel of a car body before trimming operation

In the figure final shape of part is given without trimming operation. Since trimming is not performed excess amterial cause high spring back values. This part is then trimmed to see exact spring back values.

The model generated for this part is given in figure 10. The mesh generation was performed using FEMB 28.1. Die and punch surfaces are IGES surfaces.

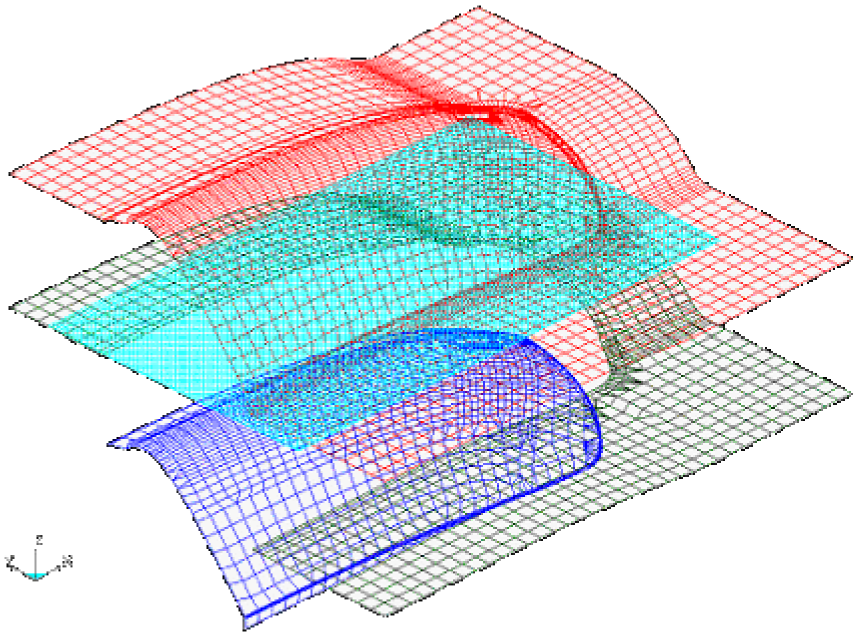

Figure 10. FEM model of Part, Die and Punch

4. Results

Many automobile panels are designed and punch ,die and draw beads are manufactured in FORM2000 Co each month. In the study the technical data for these parts are used. The part shown in figure 10 is formed and the results after forming is given in figure 11. The analyses was performed on LS-Dyna 970 3535 version solver by using IBM 330 x series 18 node super computer. In figure 11 the redundancy (change in strain) distribution is shown. This is a good distribution for such a panel. The thickness distribution on plate is given in figure 12 also. The distribution is also satisfactory.

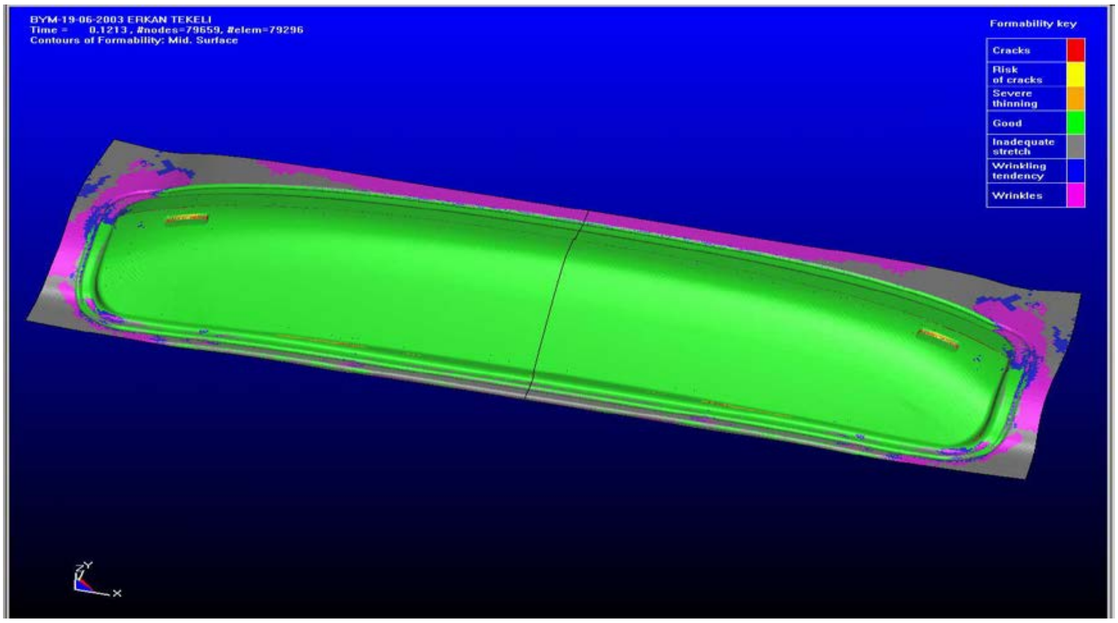

Figure 11. The redundancy distribution on the part

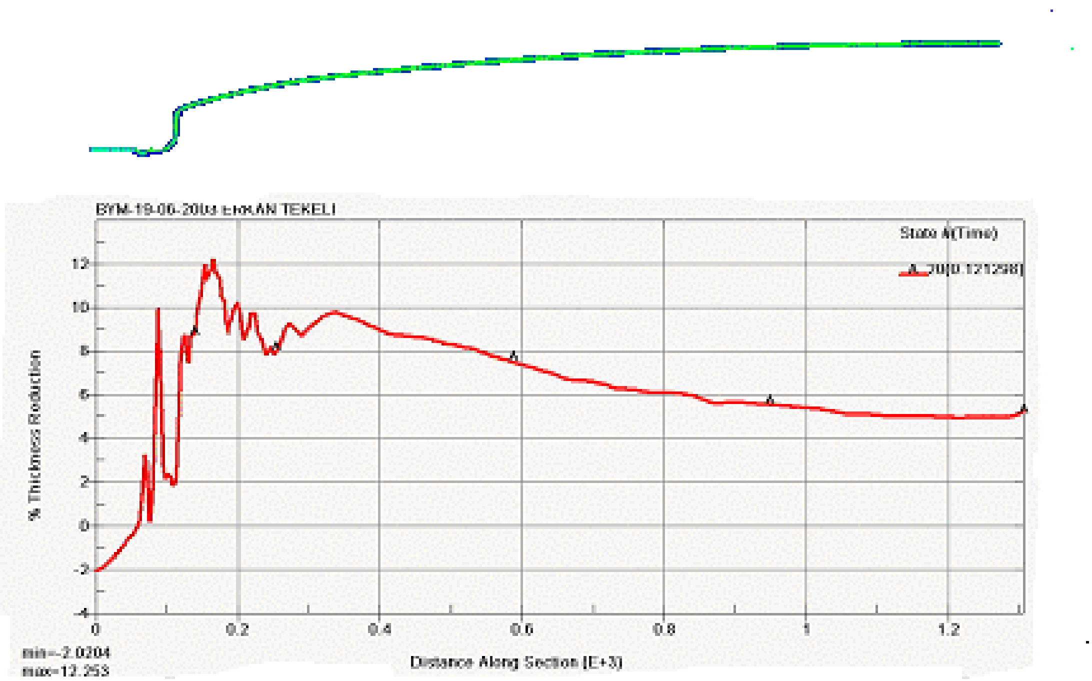

Figure 12. The thickness distribution on part after forming

When the results of forming are accepted the next step is the use of SR method to obtain surface regressions of stresses and strains. The stress and strain values obtained are used to determine the surface equations. For this pourpose, Matlab program is used and following equation is obtained.

$$\text{Stress} = -1.2553631e\text{-}005x + 4.6737364e\text{-}008x^2 - 7.2044404e\text{-}005y + 1.8291392e\text{-}006y^2 + 1.5469741e\text{-}003z - 2.8991267e\text{-}006z^2$$

After the regression method is employed, trimming of body is operated and excess material is removed. Then the new output file is written. In the new file new nodes and elements exist but their stress and strain values are zero. The trimmed part is shown in figure 13. Stress and strain values for newly formed nodes are calculated by means of equations created in Matlab.

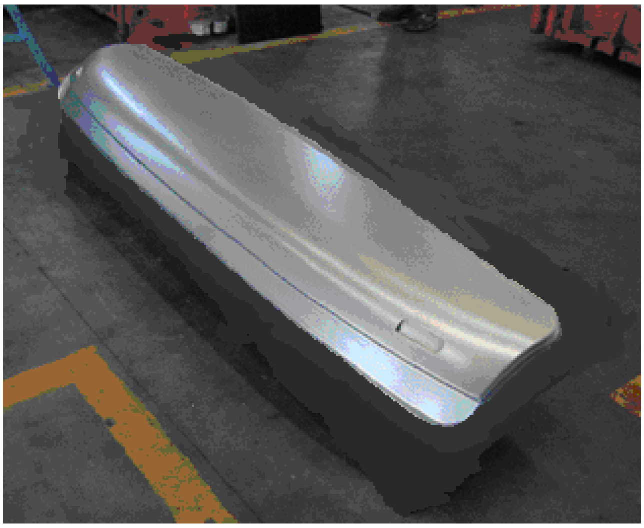

Figure 13. Trimmed body

The mapping technique presented in this study uses surface response regression method for mapping old data on the new nodes imposed due to trimming surfaces. The trimming methodology is similar to the previous studies. The differences on the results are significant because of the spring back calculations. The displacement values and directions are the only unique reason for the correctness of spring back results.

5. Conclusion

The method studied in this paper is used for reducing errors in spring back calculations. Method depends on Surface Regression of Stress and Strain data for x, y and z coordinates. The success of method was tested and experimental values are compared with the results obtained with and without SR method. The experimental values differs with the results obtained from analysis without SR method almost 4 % whereas the difference is 0.3 % with the results with SR method. The tests were repeated many times. The results are satisfactory to use this method in spring back calculations.

References

[1] Chris Galbraith, “Simulation of Sheet Metal Forming in Ls Dyna” Lecture Notes,Metal Forming Analyses Coorparation, Canada, 2001

[2] Gang Liu, Zhongqin Lin, Youxia Bao, “ Improving dimensional accuracy of a u-shaped part through an design experiment”, Finite element in analyses and design ,39, 2002 pp.107-118.

[3] Yang Yuying “Computational Sheet Metal Forming Technology” National Defense Publication PRC, 1996

[4] M. Kawka, A. Makinouchi, S.P. Wang et al, Simulating Manufacturing Processes, Automot. Eng. 105 (4) (1997) 31-33.

[5] K.Kazama, T. Nukaga, H. Makino, Spring back Simulation of Truck’s Frame Side Member, Proceeeding of Numisheet’99 Besancon, France, 1999, pp. 65-70.

[6] Y.C.Liu, The Effect of Restraining Force on Shape Deviation in Flanged Channels, ASME, J. Eng. Mater Technology 110 (1988) 389 –394.

[7] Nathan Litke, Adi Levin, and Peter Schröder, ” Trimming for subdivision surfaces”, Computer Aided Geometric Design, Volume 18, Issue 5 , June 2001, Pages 463-481.

[8] Joon C. Park and Yun C. Chung, “A tolerant approach to reconstruct topology from unorganized trimmed surfaces”, Computer-Aided Design,Volume 35, Issue 9 , August 2003, Pages 807-812.

[9] Ramon F. Sarraga, “Modifying CAD/CAM surfaces according to displacements prescribed at a finite set of points”,

Computer-Aided Design, Volume 36, Issue 4 , February 2004, Pages 343-349

[10] Kenji Shimadaa and David C. Gossard, "Automatic triangular mesh generation of trimmed parametric surfaces for finite element analysis", Computer Aided Geometric Design, Volume 15, Issue 3 , March 1998, Pages 199-222.

[11] Marco Attene, Bianca Falcidienoa, Michela Spagnuoloa and Geoff Wyvillb, "A mapping-independent primitive for the triangulation of parametric surfaces", Graphical Models, Volume 65, Issue 5 , September 2003, Pages 260-273,

[12] Mihailo Ristic, Djordje Brujic and Surya Handayani, "CAD-based triangulation of unordered data using trimmed NURBS models", Journal of Materials Processing Technology, Volume 107, Issues 1-3 , 22 November 2000, Pages 60-70.

[13] G. J. Creus, "Instability and damage effects in the modeling of metal forming", Computer Methods in Applied Mechanics and Engineering, Volume 182, Issues 3-4, 18 February 2000, Pages 421-437.

[14] Andrei V. Lyamin and Scott W. Sloan, "Mesh generation for lower bound limit analysis", Advances in Engineering Software, Volume 34, Issue 6 , June 2003, Pages 321-338.

[15] Shinobu Yoshimura, Yoshitaka Wada and Genki Yagawa, "Automatic mesh generation of quadrilateral elements using intelligent local approach", Computer Methods in Applied Mechanics and Engineering, Volume 179, Issues 1-2 , August 1999, Pages 125-138.

[16] Ch. PavanaChand and R. KrishnaKumar, "Remeshing issues in the finite element analysis of metal forming problems", Journal of Materials Processing Technology, Volume 75, Issues 1-3 , 1 March 1998, Pages 63-74.

[17] S. H. Loa, and C. K. Lee, "Selective regional refinement procedure for adaptive finite element analysis", Computers & Structures , Volume 68, Issue 4 , August 1998, Pages 325-341.

[18] D. Schmockel, M. Beth, Spring back Reduction in Draw Bending Process of Sheet Metals, An. CIRP. 42 (1) 1993 339-342.

[19] Kopp, R., Becker, M.“ A concept for dynamic remeshing at FEM-simulation showed by the example of the forging process", ICTP 90, Kyoto, Japan.