International Journal of Engineering & Technology, 7 (4.36) (2018) 1602-1607

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Rigid Plastic Finite Element Analysis of Forging Process and Its Application Using Afdex Software

A. N. Azman[1]*, N. A. Mohd Salleh[2], M. S. Joun[2], M. K. Razali[3]

[1]*Universiti Teknologi MARA Shah Alam,*
[2]*Metal Forming Research Corporation, South Korea*
[3]*Gyeongsang National University, South Korea*
**Corresponding author E-mail: amirahnabilahazman@gmail.com*

Abstract

Metal forming has been one of the important manufacturing operations in which include variety of manufacturing processes such as forging, extrusion, drawing and sheet metal. Finite Element Analysis has been used to shows whether a product will break, wear out, or work the way it was designed. However, the current forging simulator method has its limitations in providing a good result. Among the limitation are it is not accurate in the case of stress concentrations and only have capability of one stage forging process. This research has been able to introduce the basic concept of AFDEX while analyzing cold forging process of three products which are rivet nut, flat countersunk head elevator bolt and wheel lock nut in order to define the best process design and to determine the product design for each product. Several process parameters which affect the result on the simulation has been able to analyze which are type of material and type of lubricant used. Both the parameter contributed as major factor to be consider when analyzing a product. Several defects were also identified and the countermeasure were able to found during the analysis. In conclusion, this study has been able to show significance of AFDEX software as a metal forming simulator and the simulation process has able to produce the optimal die design for the three products.

Keywords: *AFDEX, finite element, metal forming, rigid plastic, forging*

1. Introduction

Metal forming is one of the most important metal processing technologies. In forging, an initially simple part, a billet for an example is plastically deformed between two tools or dies to obtain the desired final configuration [1]. The objective of analysis on metal forming processes is to provide required design information such as material flow, forming load, stress, strain, strain rate and hydrostatic pressure [2]. Finite element method has been used extensively in analyzing various metal forming processes [3, 4]. The method can provide valuable design information within a few computational minutes or hours. Hence, the number of try outs for each prototype which are time-consuming and expensive can be dramatically reduced by carrying out finite element-based computer simulation. However, it is difficult to acquire the goal of obtaining the optimal design by way of simulation, since design by simulation is basically a trial and error approach [2].

Hence, for this research, the products will be analyze using AFDEX software which is intelligent forging simulation, defined as the forging simulation for higher accuracy with minimized user intervention. AFDEX software provide more detailed description of material geometries during metal forming simulation which is vital in determining accurate simulation results. AFDEX also have capability in designing multi-stage forging processes to minimize the actual total simulation time which includes computational time and the user intervention time [5].

2. Methodology

In designing die for each product, the first step is to find out about the products that are chosen for this simulation. The required dimensions and process design of the product need to be found out first. Then, the process was proceeded to the next step which is designing die for each of the products chosen.

The process design for each product were being referred to reference books which are ASM Handbook Volume 14A Metalworking: Bulk Forming and Cold and Hot Forging Fundamentals and Applications. Next, the simulation of forging process for the products were done using AFDEX software. The designed die was saved as DXF file and then was imported into AFDEX to start the simulation process.

2.1 Input Parameter

Before running the simulation process of each product, several parameters need to be input or select into the process simulation. All the parameters include in the simulation process will be set manually.

2.1.1 Type of process, simulation, analysis and deformation

There are variety of forming process in AFDEX. This research was focussing on using forging process. The simulation was run in two dimensional and were using axisymmetric product. The unit used is in Newton and it is rigid plastic deformation.

2.1.2 Import CAD file into AFDEX

Import the die design of the products which has been drawn in AutoCAD and which was saved as DXF file.

2.1.3 Type of material used, die velocity and type of lubricant used

Choose which type of material from material library and for this research, steel AISI1015 and AISI1025 has been chosen. Next, for the velocity of the upper die, since this research is for cold forging, velocity of the upper die which is (0, -1, 0) has been input. Lastly, type of lubricant used can be either Soap Cold or Oil Cold for steel.

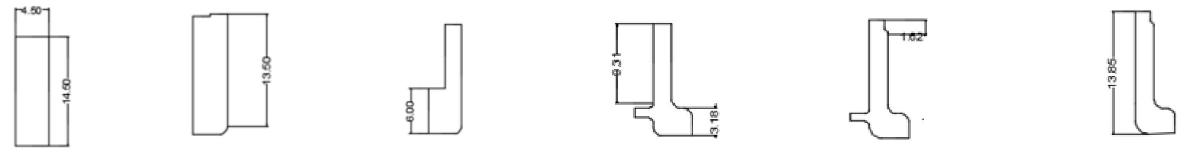
1. Result and Discussion

The important factors which need to be considered in the analysis for each product were discussed. Analysis on the parameter affecting the simulation result was done on the first product, rivet nut. Then, all the key factor that affected on the result of the simulation were being considered during the simulation on the other two products.

3.1. Rivet Nut

3.1.1 Process Design and Die Design

Based on the process design of the product 1 which is rivet nut, the die was designed and as shown on Figure 1. Figure 1 (a) is the initial length of billet for this product. For Figure 1 (b) and (c) which are stage 1 of the process, the workpiece undergoes extrusion process. The process continues with upsetting process as shown in Figure 1 (d) and (e). The last stage in Figure 1 (f) is piercing in which it is a process of indenting the surface of a work piece with a punch in order to produce hole.



(a) Initial billet | (b) First stage | (c) Second stage | (d) Third stage | (e) Fourth stage | (f) Fifth stage

Fig.1: Process Design Sequence

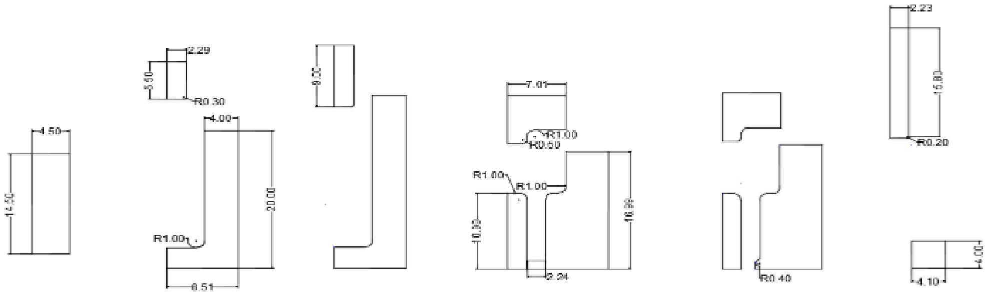


Fig. 2: Die design dimension

3.1.2 Process Configuration

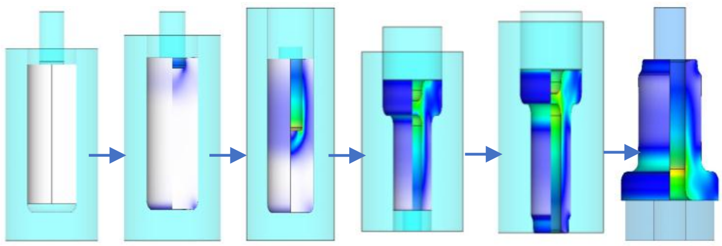
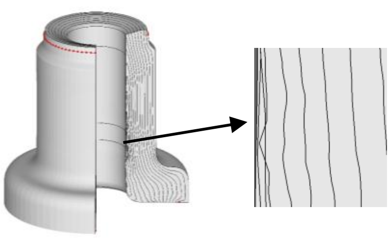


Fig. 3: Simulation Process in 3D view

3.2. Defects

3.2.1 Overlapping



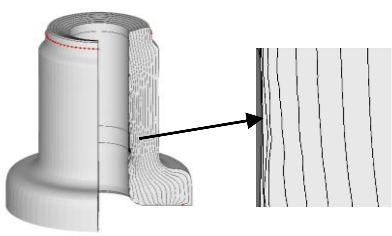


Fig. 4: Overlapping **Fig. 5:** Improvement on Overlapping

Both analysis for this product are using AISI 1015 with the same type of lubricant which is using oil. Figure 4 and Figure 5 shows the prediction of grain flow which mainly resist forces in two directions. The first one is axial direction which is tensile stress and for the horizontal direction is shear stress. The flowline of the product in Figure 4 can be observed overlapping with each other which can lead to crack at that particular area of the product. Number of elements for each stage is then increased as shown in Table 1. The result as seen in Figure 5 shows that the overlapping part on the flowline has been able to be reduced. This prove that by increasing the number of elements for meshing, the flowline of the product can be improved.

Table 1: Number of Element Before and After Improvement

	Number of Element Before Improvement	Number of Element After Improvement
Stage 1	1335	2000
Stage 2	1335	3500
Stage 3	1500	4500
Stage 4	1750	6000
Stage 5	1000	2000

3.2.1 Cracking

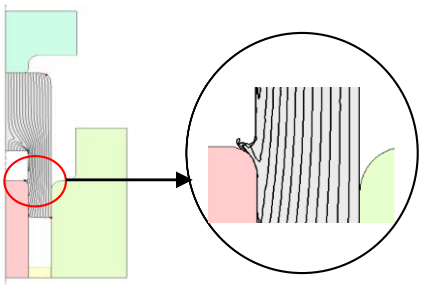


Fig. 6: Cracking

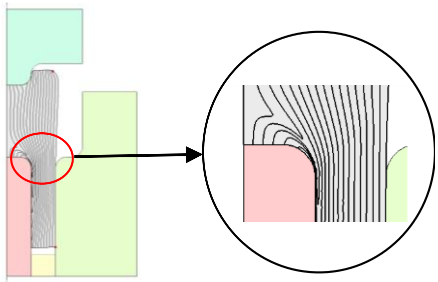


Fig. 7: After Improvement

Cracking on the material occur during the early step size on the third stage of the forming process. Based on Figure 6 and Figure 7, it is seen that there are major differences on the material flow in the die even though it was at the same stage and same step size which was step size 320. In Figure 6, the material cannot even flow into the die as the die design was too narrow too handle the excessive amount of material in the workpiece in which it is not suitable for the designed die. This situation is different compared to the material flow in Figure 7 which can flow smoothly into the die, hence, able to reduce the step size in the third stage.

By increasing the die radius from 0.4 m to 1 m, the material can flow smoothly into the die. The development of tensile stress had been able to be prevented as the material was compressed vertically at its periphery. This proves that the cause of cracking at this stage was caused by the improper die design where the material was unable to flow fully into the die.

3.3 Analysis using different type of material

3.3.1 Result when using AISI 1015

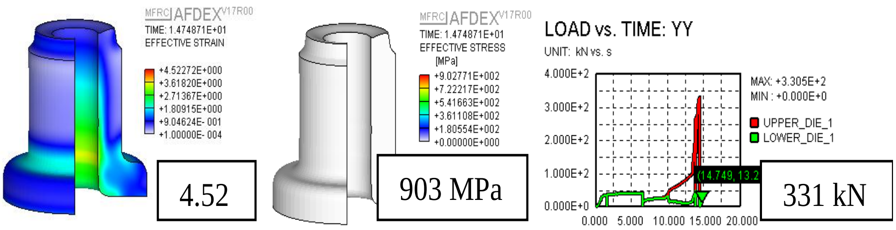


Fig. 8: Effective Strain, Effective Stress and Maximum load

3.3.2 Result when using AISI 1050

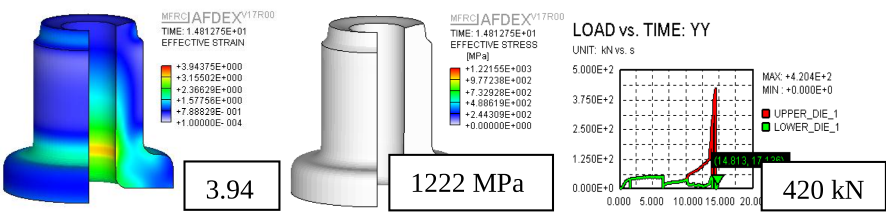


Fig. 9: Effective Strain, Effective Stress and Maximum load

Based on the result on effective stress, effective strain and the maximum load, it was shown that by using material AISI 1015, the value for each result is lower than when using material AISI 1050. For the product that was analysed with AISI 1050, the result is 1222 MPa. The result had higher value compared to when the product was analysed using AISI 1015 which was 903 MPa. For the maximum load, analysis using AISI 1015 can be seen have much lower value which is 332.2 kN rather than 420 kN when using AISI 1050. This prove that selection of material also had an effect on the result of analysis. Overlapping occur in flowline of the material using AISI 1050 which can cause defect such as cracking during the forging process.

3.4. Analysis Using Different Type of Lubricant

3.4.1. Soap

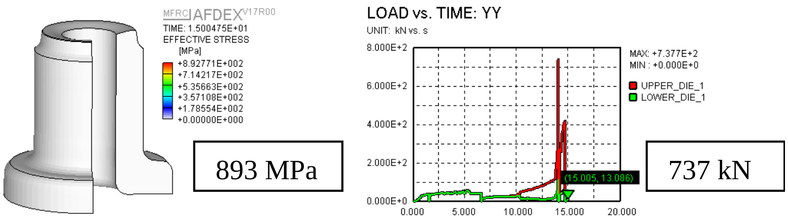


Fig. 10: Effective stress and forming load

3.4.2 Oil

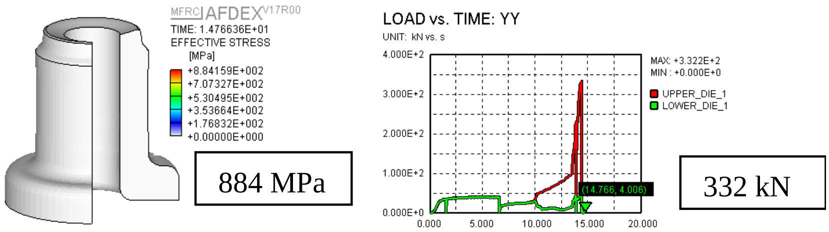


Fig. 11: Effective stress and forming load

By comparing the result for both type of lubricant referring to Figure 10 and Figure 11, it can be indicated that the forming load and stress is higher when soap is being used as lubricant. This is due to higher value of friction coefficient for soap which is 0.1 compared to oil which is only 0.025. The forming load of 332 kN was produced when oil was used which was considered low value compared to forming load of 737 kN when soap was being employed. It has been evaluated from the results that the forming load was found to be proportional to the friction coefficients which shows that as the friction values increased, the die life decreased. This is because, higher friction coefficients required higher load to fill the die cavity. Next, from the analysis, the highest effective stress for oil is 884 MPa which was slightly low than 893 MPa for soap. This shows that the stress value has increased as the friction coefficient increased, in which an increase in die wear was evaluated. Hence, the fatigue life would be reduced and subsequently, for oil lubricant, the stress concentration at punch would be decreased.

Based on the analysis on factor affecting the result of the simulation, for rivet nut, AISI 1015 had been chosen as type of material to be used while oil as the type of lubricant used for the simulation on this product. The best result after applying all the important factor which may affect the result of the analysis was presented in Table 2.

Table 2: Rivet nut final result analysis

	Effective strain	Effective stress (MPa)	Maximum load (kN)
AISI 1015, Oil	4.52	903	331

3.5 Flat Countersunk Head Elevator Bolt

The other two products are flat countersunk head elevator bolt for product two while for product three is wheel lock nut. The same type of analysis had been done for both products.

The same process had been repeated and the analysis for each had been performed. The same countermeasure was taken for the defects that had been identified before in the first product.

3.5.1 Process Design and Die Design

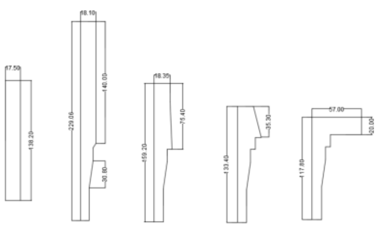


Fig. 12: Process Design

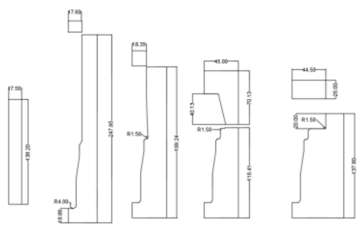


Fig. 13: Die Design Dimension

3.5.2 Material and Die Information

Table 3: Material and Die Information

Information about material	**Information about die**
Dimension: 17.5 x 138.2 mm	Friction: Oil cold (steel)
Initial temperature: Room temperature	Die velocity: Constant (Upper die: -1.0 mm/s, Lower die: 0.0 mm/s)

3.5.3 Result

Table 4: Result comparison on three types of material

	Effective strain	**Effective stress (MPa)**	**Maximum load (kN)**
AISI 1010	3.70	966	44050
AISI 1015	3.53	880	56750
AISI 1025	3.28	939	12480

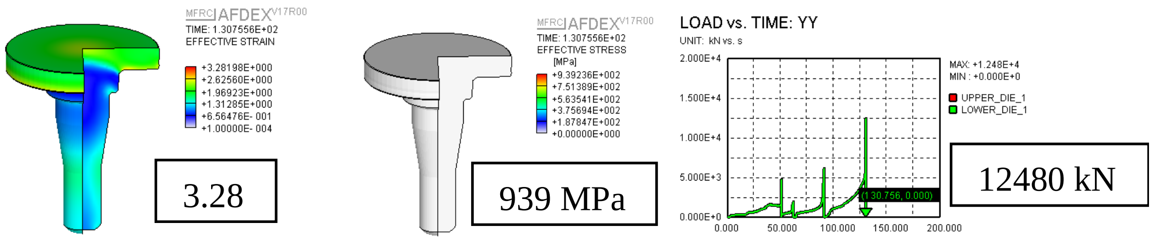


Fig. 14: Effective Strain, Effective Stress and Maximum load

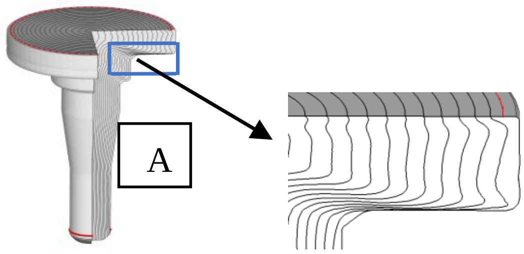


Fig. 15: Flowline

Based on the result obtained as shown in Table 4, it shows that simulation using AISI 1025 had the best result as it has the lowest maximum value for the three important parameter that need to be observed in forging process. Even though the effective stress for AISI 1015 is slightly lower than AISI 1025, the maximum load obtained for AISI 1015 had a huge difference result compared to AISI 1025. Hence, result using AISI 1025 was considered the best result for this product. Figure 14 shows the analysis when AISI 1025 was used. The flowline at position A may be analysed as a critical position where breaking may occur if high tensile stress exists in this section. Hence, any other large force need to be avoid from being applied on this part. Mild flow lines with less bending can be seen at the head part of the bolt which is beneficial to the component properties.

3.6 Wheel Lock Nut

3.6.1 Process Design and Die Design

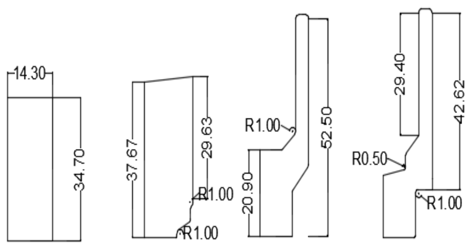


Fig. 16: Process Design

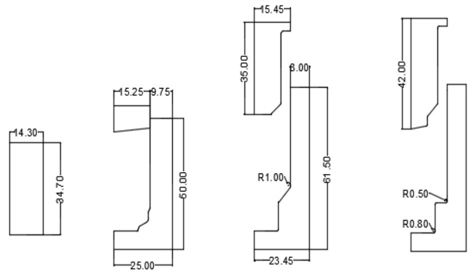


Fig. 17: Die Design Dimension

3.6.2 Material and Die Information

Table 5: Material and Die Information

Information about material	Information about die
Dimension: 14.3 x 34.7 mm	Friction: Soap cold (steel)
Initial temperature: Room temperature	Die velocity: Constant (Upper die: -1.0 mm/s, Lower die: 0.0 mm/s)

Table 6: Result comparison on three types of material

	Effective strain	Effective stress (MPa)	Maximum load (kN)
AISI 1010	3.95	1140	3614
AISI 1015	4.18	897	2578
AISI 1025	4.39	1165	3240

3.6.3 Result

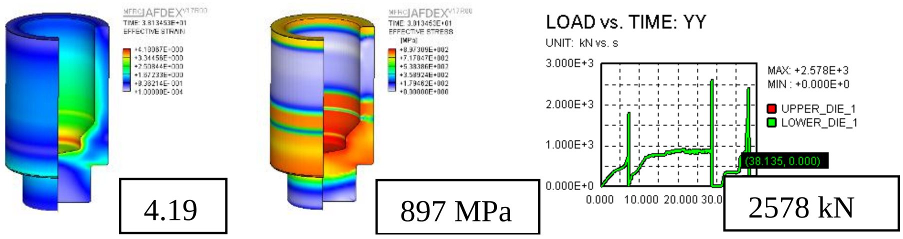


Fig. 18: Effective Strain, Effective Stress and Maximum load

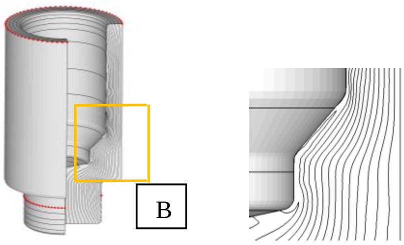


Fig. 19: Flowline

As shown in Table 6, by comparing with the other two type of material, the simulation using AISI 1015 had produced lowest value of maximum effective stress and maximum load which are 897 MPa and 2578 kN respectively. The result on the simulation can be seen in Figure 18. Material flow of this product was shown in Figure 19. In order to form the circular inner diameter, higher force is needed to form the desired shape. Hence, the position at B has been predicted to be where the maximum stress will occur. The result on the effective stress also shows that maximum stress was predicted to occur at the bottom part of the product in which position B is a part of the predicted area.

3.7 Application in Industry

All three products above plays an important part in the industry. For rivet nut, it is used in various industry such as automotive and aviation. In automotive industry, it is used in mounting for bumper and frame system of a car. Rivet nut are designed to provide strong threads in thin materials.

Next, flat countersunk head elevator bolts was originally designed for the attaching of elevator buckets to a belt used in the vertical conveying of bulk materials. In this current day, these bolts are widely used in a variety of applications unrelated to conveyor systems. Examples of its application in the industry are they secure flooring and also used as levelling legs. This type of bolt can be found on snowmobiles and skateboards. It is the main component to be used wherever the head clearance is crucial.

Lastly, wheel lock nut is commonly used in automotive industry. It is used to secure a wheel on a vehicle in which they are designed to prevent thieves from stealing rims off from a car. It have a specific pattern for the nut that only the key that comes with the set can install or remove them.

4. Conclusion

As the conclusion for the study carried out on the three products, the objective of this study had been able to be achieved. The basic concept of AFDEX as a metal forming simulator had been introduced in this study. AFDEX software have the capability of analysing the product using multistage process. This helps minimize simulation time allocated to analyse a product. During the research, the flexibility of this software had been proven as it was able to perform analysis using different type of material, lubricant and meshing properties. A convenient and user-friendly simulation had been demonstrated throughout this research in which the analysis process for each product had been able to be handle easily.

This study had been able to analyse several parameters which affected the result for the simulation. Selection of material is one of the factors which had been proven affecting the result. Type of material chosen need to include the characteristics of the product which was being analysed.

Next, the effect on lubrication selection was observed through this research as it shows that the maximum load is higher when the coefficient of friction of a material is higher also had an effect on the result of analysis the simulation. Defects in forging process which were cracking and overlapping were also identified and the countermeasure were able to be perform during the analysis of the product.The best process design and product design for each product using CAD tools has been able to define in this study. Analysis on each of the parameter are the key factor in determining the optimal die design for each product.

For future research, other type of bulk forming processes in AFDEX software such as extrusion, drawing and plate forging can be analyse using AFDEX.

Acknowledgement

This study was conducted with the support of Professor Man Soo Joun, President of Metal Forming Research Corporation and Research lab team at Gyeongsang National University, South Korea.

References

[1] Taylan Altan, Gracious Ngaile, Gangshu Shen, “Cold and Hot Forging: Fundamentals and Applications”, ASM International, United States of America (2004)

[2] Man Soo Joun, “Process Optimal Design in Steady-State Metal Forming by Finite Element Method”, Pohang,Korea (1992)

[3] O. C. Zienkiewicz, “The Finite Element Method”, McGraw-Hill (1977)

[4] S. Kobayashi, S. I. Oh, T. Altan, “Metal Forming and The Finite-Element Method”, Oxford University Press, New York (1989)

[5] Man-Soo Joun, Min-Cheol Lee, Jae-Gun Eom, Qiu-Shi Li, Shang-Hyun Shim, “An Intelligent Metal Forming Simulator AFDEX and its Applications”, International Symposium on Mechanics, Aerospace and Informatics Engineering, ISMAI04-MF-01 (2009)