International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-6 Issue-6, August 2017

Effect of Friction on Extrusion of Non-Ferrous Alloys- Modelling and Simulation

K. Kiran Kumar, S. Suresh Kumar

Abstract: ***Extrusion is a process used to create jobs of a fixed cross sectional profile. A material is pushed or drawn through a die of the desired cross section in this particular problem the radius of an Aluminum cylindrical bar is reduced 33% by an extrusion process. The generation of heat due to plastic dissipation in side the bar and the frictional heart generation at the work piece/die inter face are considered. This analysis has been performed in ABACUS/standard. A fully coupled temperature- displacement analysis is performed with the die kept at a constant temperature. In ABAQUS/standard the die is modeled with CAX4T elements made into an iso thermal rigid body using the *RIGID BODY, ISO THERMAL option are made with an analytical rigid surface. The results obtained with ABAQUS/standard. After analyzing and simulating on two different materials ALUMINIUM & COPPER it is observed that copper offers better extrusion capabilities as compared to aluminum.***

Keywords: ABAQUS, CAX4T, ISO, RIGID BODY, ISO THERMAL, ALUMINIUM & COPPER, ABAQUS

I. INTRODUCTION

1.1. Manufacturing Process

The knowledge of manufacturing process is of great importance for a design engineer. The manufacturing process contains a few steps that are important in reducing waste and reducing engineering flaws before proceeding. The process varies by manufacturer but is crucial to the overall cost of a part or product. Preliminary tests must be performed with a manufacturing process to ensure that the product or part being made is engineered correctly and that all flaws are found before the product or part reaches the final stages. Much of this analysis takes place with the CAM, or computer-aided manufacturing, program. An important step in a manufacturing process is to create a prototype. Prototype machining is often done in a less expensive material to cut down on costs, and rapid prototyping is often done on a 3D printer, which creates the part from the CAM program. This step allows engineers to study the prototype and expose flaws in design. The main step in the manufacturing process is the creation of the part in the proper material and starting a production run to make multiple parts with accuracy and consistency using a CNC, or computer numerical control, machine, such as a CNC lathe, miller or router.

Manuscript published on 30 August 2017.
***** Correspondence Author (s)
Mr. K. Kiran Kumar, (Ph.D), Asst. Professor, Department Mechanical Engineering, AVN Institute of Engineering & Technology, Ramdas Pally (Telangana)-501510, India. E-mail: kirankrnt@gmail.com
Mr. S. Suresh Kumar, Asst. Professor, Department Mechanical Engineering, Kshatriya College of Engineering, Nirmal (Telangana)-501510, India. E-mail: sopparisuresh@gmail.com

The totality of the design & manufacturing process is defined by implementation, and differs in detail within every manufacturing company. The following is representative of that process as viewed from within the turnkey industry.

1.2. Extrusion

The process of extrusion is simply forcing a billet of metal through a shaped die to produce a continuous length of constant section similar to the die profile. There are two basic extrusion processes shown in Fig.1.2, direct extrusion and indirect extrusion. Direct extrusion is by far the most widely used process. Indirect extrusion more efficient and produced higher quality products.

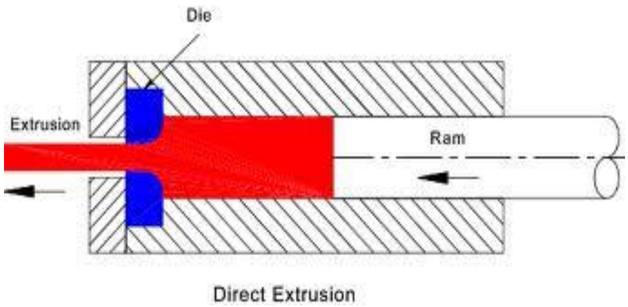


Fig 1: Direct Extrusion

Extrusion is a process used to create objects of a fixed cross-sectional profile. A material is pushed or drawn through a die of the desired cross-section. The two main advantages of this process over other manufacturing processes are its ability to create very complex cross-sections and work materials that are brittle, because the material only encounters compressive and shear stresses. It also forms finished parts with an excellent surface finish. Extrusion may be continuous (theoretically producing indefinitely long material) or semi-continuous (producing many pieces). The extrusion process can be done with the material hot or cold.

II. OBJECTIVES

The basic objective of the present study is to simulate the extrusion process considering the heat generation due to plastic deformation and frictional heat generation at the work piece /die interface. The radius of an aluminum cylindrical bar is reduced 33% by an extrusion process in a single stroke. And study the effect of friction on the equivalent plastic strain and stresses in the material. And compare the extrusion behavior of two different materials such as aluminum and copper.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

III. PROBLEM DEFINITION:

In this particular problem the radius of an aluminum cylindrical bar is reduced 33% by an extrusion process. The generation of heat due to plastic dissipation inside the bar and the frictional heat generation at the work piece/die interface are considered.

3.1. Future Scope of Work:

- The material is hardens isotropically.
- The strain rate dependence is ignored.

3.2. Brief Introduction To Software Used:

ABAQUS FEA (formerly **ABAQUS**) is a suite of software applications for finite element analysis and computer aided engineering. The name of this software is derived from the Greek word, "abax" meaning "board covered with sand"

IV. LITERATURE REVIEW:

Several research works reported in literature on FEA of extrusion process. The brief review is reported in the following programs.

The multi-hole extrusion of aluminum-alloy tubes was examined by the **Fuh-Kuo Chena, Wen-Chan Chuang, Shan Torng[1]**. in the present study using both the finite element analysis and the experimental approach. The finite element analysis was first validated qualitatively and quantitatively by the experimental data obtained from the single-hole extrusion process. The effects of the process parameters, such as extrusion temperature, extrusion speed, dimensions of billet, and location of holes on the extrusion load and the shape of extruded tubes were then studied by the finite element analysis.

The aim of this paper is to study the effect of extrusion parameters (extrusion speed and temperature) and die geometry, i.e. extrusion radius, on the extruded billet quality (Equivalent stress and strain) using F.E.M. technique. Was studied by **A.E. Lontos, F.A. Soukatzidis, D.A. Demosthenous, A. K. Baldoukas [2]** For this purpose the general F.E.A. software Deform-2D has been used to set up the finite element model of the warm aluminum extrusion in two dimensions (2D)

***P. Tiernan, M.T. Hillery, B.Draganescu, M. Gheorghe*[3]** reports on an experimental and finite element analysis (FEA) of the cold extrusion of high-grade (AA1100) aluminium. The influence of die angle, reduction ratio and die land on the extrusion force during the extrusion process was investigated

***I.Flitta and T. Sheppard* [4]** This investigation focuses on simulation of the extrusion process and in particular the effect of the initial billet temperature on friction and its consequences on material flow.

V. FINITE ELEMENT METHOD:

5.1. Introduction:

Finite element analysis seeks to approximate the behavior of an arbitrarily shaped structure under general loading and constraint conditions with an assembly of discrete finite elements. Finite elements have regular geometric shapes and known solutions. The behavior of the structure is obtained by analyzing the collective behavior of the elements.

5.2. Engineering Analysis:

Engineering analysis can be broadly divided into two categories.

1. Classical methods and
2. Numerical methods.

5.3. Types of Analysis Used in Structural Analysis

- Linear static analysis
- Nonlinear static analysis

5.3.1. Linear "Static Analysis"

Static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by the loads.

Overview

A static stress analysis:

- Is used when inertia effects can be neglected;
- Can be linear or nonlinear; and
- Ignores time-dependent material effects (creep, swelling, viscoelasticity) but takes rate-dependent plasticity and hysteretic behavior for hyper elastic materials into account

Time period

During a static step you assign a time period to the analysis. This is necessary for cross-references to the amplitude options, which can be used to determine the variation of loads and other externally prescribed parameters during a. In some cases this time scale is quite real –for example, the response may be cause by temperatures varying with time base on a previous transient heat transfer run; or the material response may be rate dependent (rate-dependent plasticity), so that a natural time scale exists. Other case does not have such a natural time scale; for example, when a vessel is pressurized up to limit loads with rate-independent material response. If you don't specify a time period, Nx-Nastran defaults to a time period in which "time "varies from 0.0 to 1.0 over the step. The "time" increments are then simply fractions of the total period of the step.

Linear Static analysis

Linear static analysis involves the specification of load cases and appropriate boundary conditions. If all or part of a problem has linear response, sub structuring is a powerful capability for reducing the computational cost of large analysis.

5.3.2. Non Linear Static Analysis:

Non-linearity can arise from large-displacement effects, material nonlinearity, and/or boundary nonlinearities such as contact and must be accounted for. If geometrically nonlinear behavior is expected in a step, the large-displacement formulation should be used. In most nonlinear analyses the loading variations over the step follow a prescribed history such as a temperature transient or a prescribed displacement.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-6 Issue-6, August 2017

Linear static Analysis process

In static analysis, adequate boundary conditions must be applied to the model in order to prevent any rigid body motion of the structure. If the specified boundary conditions do not adequately constrain the model in all direction the structure's stiffness matrix remains singular and the run terminates with an error message. Once the boundary conditions are applied to the model appropriately, the global stiffness matrix is reduced to a non-singular stiffness matrix representing the constrained structure. All of the loads that you apply to the model are combined to form the load vector. These applied loads can be in the form of point forces and moments applied directly to the grid points, line loads applied along the length of one-dimensional elements, surface loads applied to two- and three dimensional elements, or body loads such as gravity. These different load types may be combined to form a single load vector, which is the same as saying that the loads are applied simultaneously. Also there is option of applying multiple load vectors within a single run. After the constrained stiffness matrix and the load vector are generated, the static equilibrium.

The actual solution stage involves solving the matrix equations for the nodal displacements. For a static stress analysis, the equation to the solved is of the form.

[K][U] =[F]

Where, [K] =Stiffness matrix

[U]=Nodal Displacements

[F]=Nodal forces

The solution to this set of equation is obtained by Gauss elimination procedure or some derivative of this method. The final step in this process is to calculate the stress involved with the deflections in the structure. Once the displacements at the grid points are known, any desired outputs, such as element forces, strains, and stresses, are computed.

VI. FINITE ELEMENT MODELING AND ANALYSIS OF EXTRUSION OF A CYLINDRICAL METAL BAR WITH FRICTIONAL HEAT GENERATION FINITE ELEMENT MODELING PROCEDURE

The purpose of finite element modeling is to make the model that behaves mathematically, like the structure that is being modeled. Once the geometric modeling is completed according to dimension, the model is converted in to a finite element (FE) model. The first step of the finite element method is the replacement of the complex system in to equivalent idealized one consisting of individual elements connected to each other at specified points or nodes, elements, physical and material properties, loads and boundary conditions.

6.1. Problem Definition

This analysis illustrates how extrusion problems can be simulated with ABAQUS. In this particular problem the radius of an aluminum cylindrical bar is reduced 33% by an extrusion process. The generation of heat due to plastic dissipation inside the bar and the frictional heat generation at the workpiece/die interface are considered.

ABAQUS/Standard, a general-purpose finite element program is used in the simulation of extrusion of a cylindrical metal bar with frictional heat generation

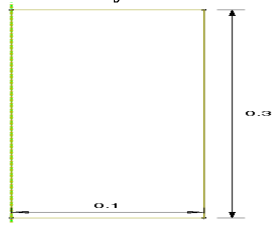
6.1.1. Finite Element Modeling

The analysis has been performed in ABAQUS/Standard and ABAQUS/Explicit, heat transfer between the deformable bar and the rigid die is not considered, although frictional heating is included. A fully coupled temperature-displacement analysis is performed with the die kept at a constant temperature. In addition, an adiabatic analysis is presented using ABAQUS/Standard without accounting for frictional heat generation. Both the node-to-surface (default) and the surface-to-surface contact formulations in ABAQUS/Standard are presented. The surface-to-surface contact formulation is invoked by using the TYPE=SURFACE TO SURFACE parameter on the CONTACT PAIR option. Various techniques are used to model the rigid die. In ABAQUS/Standard the die is modeled with CAX4T elements made into an isothermal rigid body using the *RIGID BODY, ISOTHERMAL option and with an analytical rigid surface.

6.2. FEA of Extrusion of a Cylindrical Metal bar With Frictional Heat Generation

Analysis illustrates how extrusion problems can be simulated with ABAQUS. In this particular problem the radius of an aluminum cylindrical bar is reduced 33% by an extrusion process. The generation of heat due to plastic dissipation inside the bar and the frictional heat generation at the work piece/die interface are considered. Primary analysis in both ABAQUS/Standard and ABAQUS/Explicit, heat transfer between the deformable bar and the rigid die is not considered, although frictional heating is included. A fully coupled temperature-displacement analysis is performed with the die kept at a constant temperature. In addition, an adiabatic analysis is presented using ABAQUS/Standard without accounting for frictional heat generation. Both the node-to-surface (default) and the surface-to-surface contact formulations in ABAQUS/Standard are presented. The surface-to-surface contact formulation is invoked by using the TYPE=SURFACE TO SURFACE parameter on the CONTACT PAIR option. Various techniques are used to model the rigid die. In ABAQUS/Standard the die is modeled with CAX4T elements made into an isothermal rigid body using the *RIGID BODY, ISOTHERMAL option and with an analytical rigid surface.

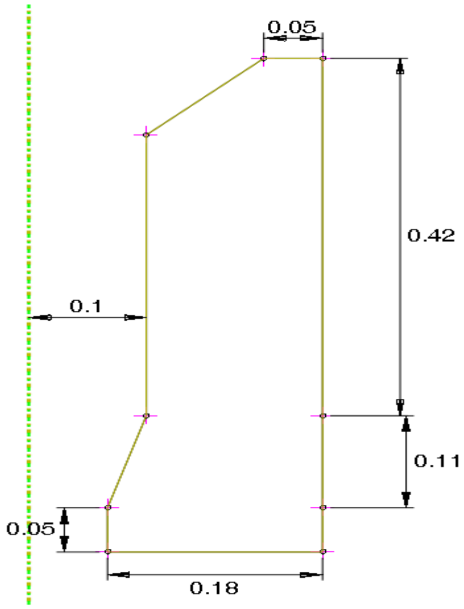
Geometry and Model



Model of Billet

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

All Dimensions Are in m



Model of Die and container

Figure 4.1 Geometric model

The bar has an initial radius of 100 mm and is 300 mm long. Figure 4.1 shows half of the cross-section of the bar, modeled with first-order axisymmetric elements (CAX4T and CAX4RT elements in ABAQUS/Standard and CAX4RT elements in ABAQUS/Explicit). In the primary analysis in both ABAQUS/Standard and ABAQUS/Explicit, heat transfer between the deformable bar and the rigid die is not considered, although frictional heating is included. A fully coupled temperature-displacement analysis is performed with the die kept at a constant temperature. In addition, an adiabatic analysis is presented using ABAQUS/Standard without accounting for frictional heat generation. Both the node-to-surface (default) and the surface-to-surface contact formulations in ABAQUS/Standard are presented. The surface-to-surface contact formulation is invoked by using the TYPE=SURFACE TO SURFACE parameter on the *CONTACT PAIR option. In the case of ABAQUS/Explicit, penalty and kinematic contact formulations are used in the definition of contact interactions. Various techniques are used to model the rigid die. In ABAQUS/Standard the die is modeled with CAX4T elements made into an isothermal rigid body using the *RIGID BODY, ISOTHERMAL option and with an analytical rigid surface. In ABAQUS/Explicit the die is modeled with an analytical rigid surface and discrete rigid elements (RAX2). The FILLET RADIUS parameter on the *SURFACE option is set to 0.075 for models using an analytical rigid surface to smoothen the die surface. The ABAQUS/Explicit simulations are also performed with Arbitrary Lagrangian-Eulerian (ALE) adaptive meshing and enhanced hourglass control as shown in Figure 4.2.

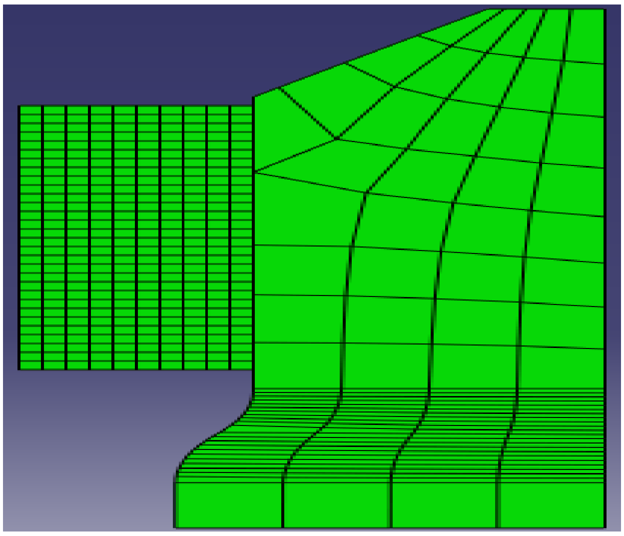


Figure 4.2 Assembly Model

6.3. Material Model and Interface Behavior

Table 4.1 Material Model

Name	Young's Modulus	Poisson' S Ratio
Aluminium	69 GPa	0.33
Copper	110 GPa	0.32

The material model is chosen to reflect the response of a typical commercial purity aluminum alloy. The material is assumed to harden isotropically and the materials properties are shown in Table 4.1. The dependence of the flow stress on the temperature is included, but strain rate dependence is ignored. Instead, representative material data at a strain rate of 0.1 sec^{-1} are selected to characterize the flow strength.

The interface is assumed to have no conductive properties. Coulomb friction is assumed for the mechanical behavior, with a friction coefficient of 0.1. The *GAP HEAT GENERATION option is used to specify the fraction, f_g, of total heat generated by frictional dissipation that is transferred to the two bodies in contact. Half of this heat is conducted into the work piece, and the other half is conducted into the die. Furthermore, 90% of the non-recoverable work due to plasticity is assumed to heat the work material.

Boundary conditions, loading, and solution control: In the first step the bar is moved to a position where contact is established and slipping of the work piece against the die begins. In the second step the bar is extruded through the die to realize the extrusion process. This is accomplished by prescribing displacements to the nodes at the top of the bar. In the third step the contact elements are removed in preparation for the cool down portion of the simulation. In ABAQUS/Standard this is performed in a single step: the bar is allowed to cool down using film conditions, and deformation is driven by thermal contraction during the fourth step. Volume proportional damping is applied to two of the analyses that are considered.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

In one case the automatic stabilization scheme with a constant damping factor is used.

A non-default damping density is chosen so that a converged and accurate solution is obtained. In another case the adaptive automatic stabilization scheme with a default damping density is used. In this case the damping factor is automatically adjusted based on the convergence history.

In ABAQUS/Explicit the cool down simulation is broken into two steps: the first introduces viscous pressure to damp out dynamic effects and, thus, allow the bar to reach static equilibrium quickly; the balance of the cool down simulation is performed in a fifth step. The relief of residual stresses through creep is not analyzed in this example. In ABAQUS/Explicit mass scaling is used to reduce the computational cost of the analysis; non-default hourglass control is used to control the hour glassing in the model. The default integral viscoelastic approach to hourglass control generally works best for problems where sudden dynamic loading occurs; a stiffness-based hourglass control is recommended for problems where the response is quasi-static. A combination of stiffness and viscous hourglass control is used in this problem. For purposes of comparison a second problem is also analyzed, in which the first two steps of the previous analysis are repeated in a static analysis with the adiabatic heat generation capability. The adiabatic analysis neglects heat conduction in the bar. Frictional heat generation must also be ignored in this case. This problem is analyzed only in ABAQUS/Standard and the meshing property and geometry model are shown in Figure 4.3.

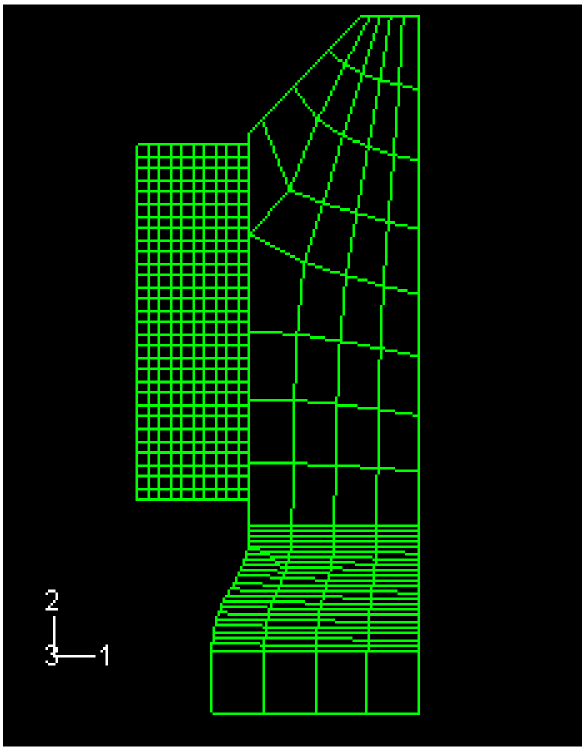


Figure 4.3 Axisymmetric extrusions with meshed rigid die, ABAQUS/Standard

VII. RESULTS AND DISCUSSION

The following discussion centers around the results obtained with ABAQUS/Standard. The results of the ABAQUS/Explicit simulation are in close agreement with those obtained with ABAQUS/Standard for both the node-to-surface and surface-to-surface contact formulations.

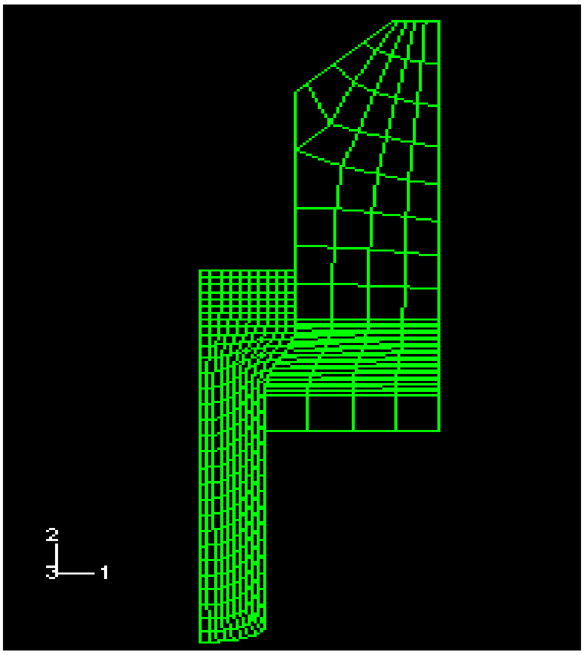


Figure 5.1 Deformed configurations

Figure 5.1 shows the deformed configuration after Step 2 of the analysis. Figure5.2 and Figure 5.3 show contour plots of plastic strain and the Mises stress at the end of Step 2 for the fully coupled analysis using CAX4RT elements. These plots show good agreement between the results using the two contact formulations in ABAQUS/Standard. The plastic deformation is most severe near the surface of the workpiece, where plastic strains exceed 100%. The peak stresses occur in the region where the diameter of the workpiece narrows down due to deformation and also along the contact surface. Figure 5.4 compares nodal temperatures obtained at the end of Step 2 using the surface-to-surface contact formulation in

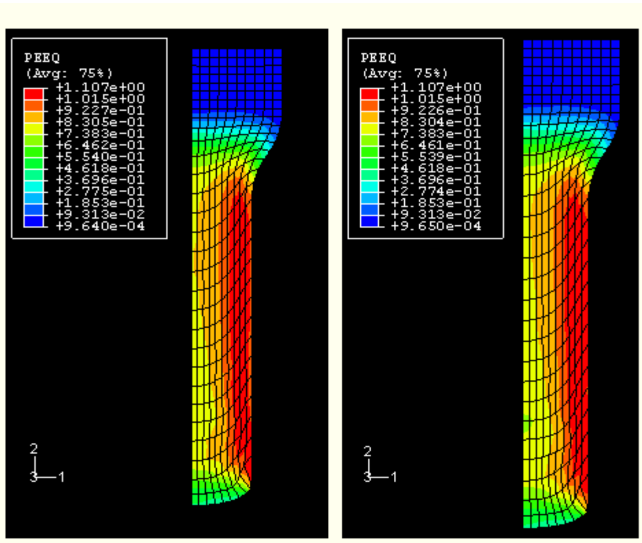


Figure 5.2 Plastic Strain Contours: Step 2, Thermally Coupled Analysis

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

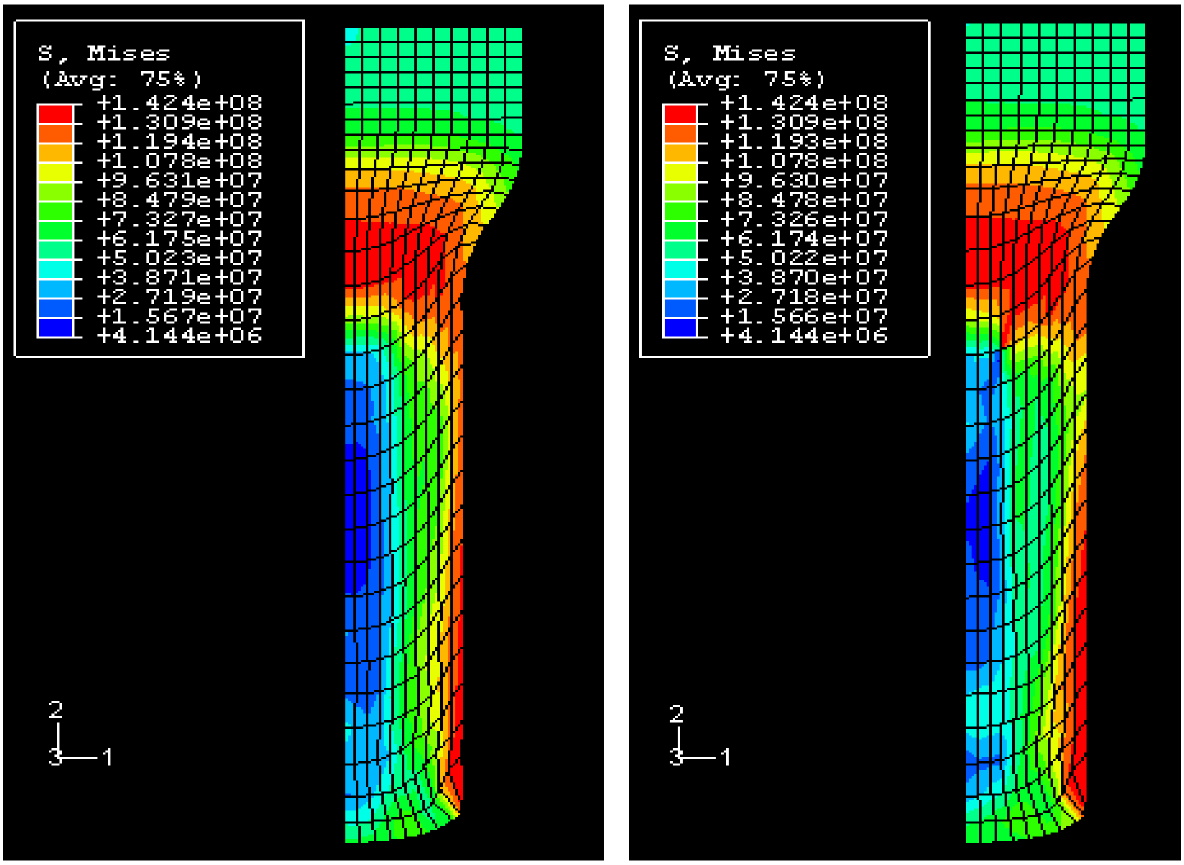


Figure 5.3 Mises stress contours: Step 2, thermally coupled analysis

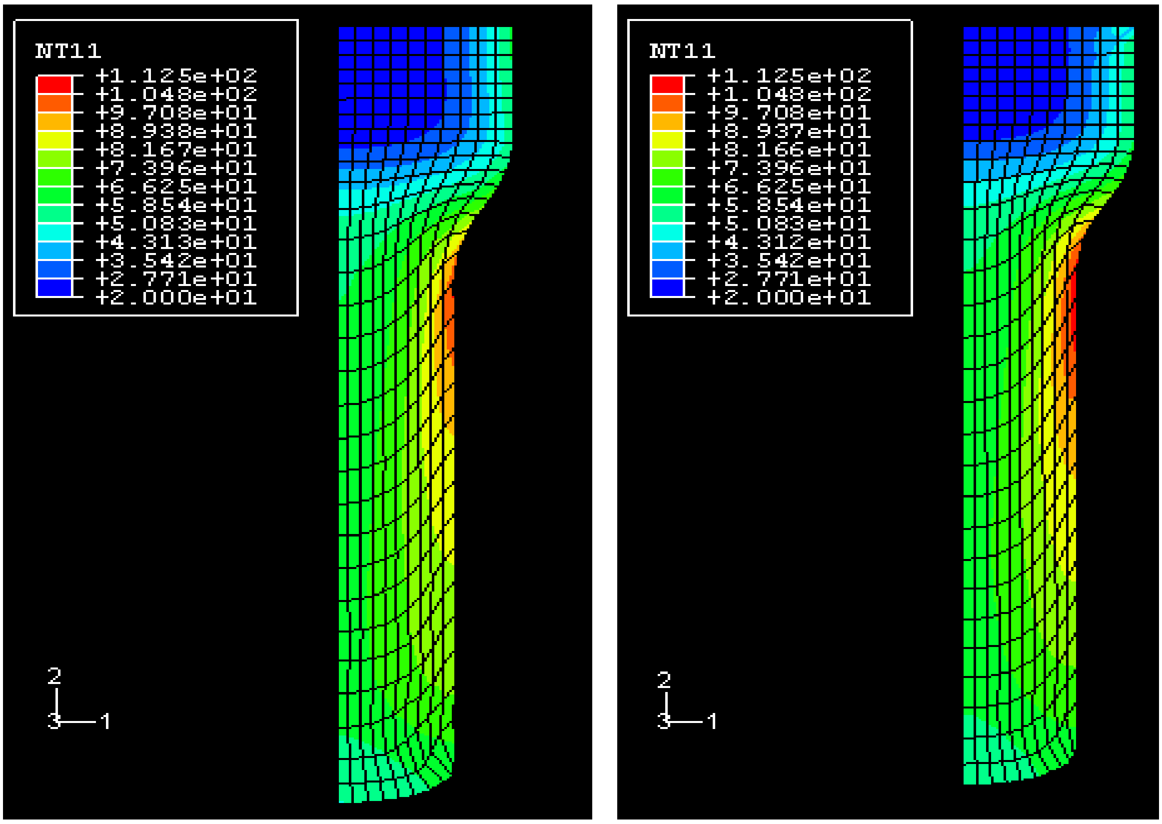


Figure 5.4 Temperature contours: Step 2, thermally coupled analysis frictional heat generation); surface-to-surface contact formulation in ABAQUS/Standard, left; ABAQUS/Explicit, right.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

ABAQUS/Standard with those obtained using kinematic contact in ABAQUS/Explicit. In both cases CAX4RT elements are used. The results from both of the analyses match very well even though mass scaling is used in ABAQUS/Explicit for computational savings. The peak temperature occurs at the surface of the workpiece because of plastic deformation and frictional heating. The peak temperature occurs immediately after the radial reduction zone of the die. This is expected for two reasons. First, the material that is heated by dissipative processes in the reduction zone will cool by conduction as the material progresses through the post reduction zone. Second, frictional heating is largest in the reduction zone because of the larger values of shear stress in that zone.

Similar results were obtained with the two types of stabilization considered. Adaptive automatic stabilization is generally preferred because it is easier to use. It is often necessary to specify a non default damping factor for the stabilization approach with a constant damping factor; whereas, with an adaptive damping factor, the default settings are typically appropriate.

Figure 5.5 compares results of a thermally coupled analysis with an adiabatic analysis using the surface-to-surface contact formulation in ABAQUS/Standard. If we ignore the zone of extreme distortion at the end of the bar, the temperature increase on the surface is not as large for the adiabatic analysis because of the absence of frictional heating. As expected, the temperature field contours for the adiabatic heating analysis, shown in Figure 5.5, are very similar to the contours for plastic strain from the thermally coupled analysis, shown in 5.2

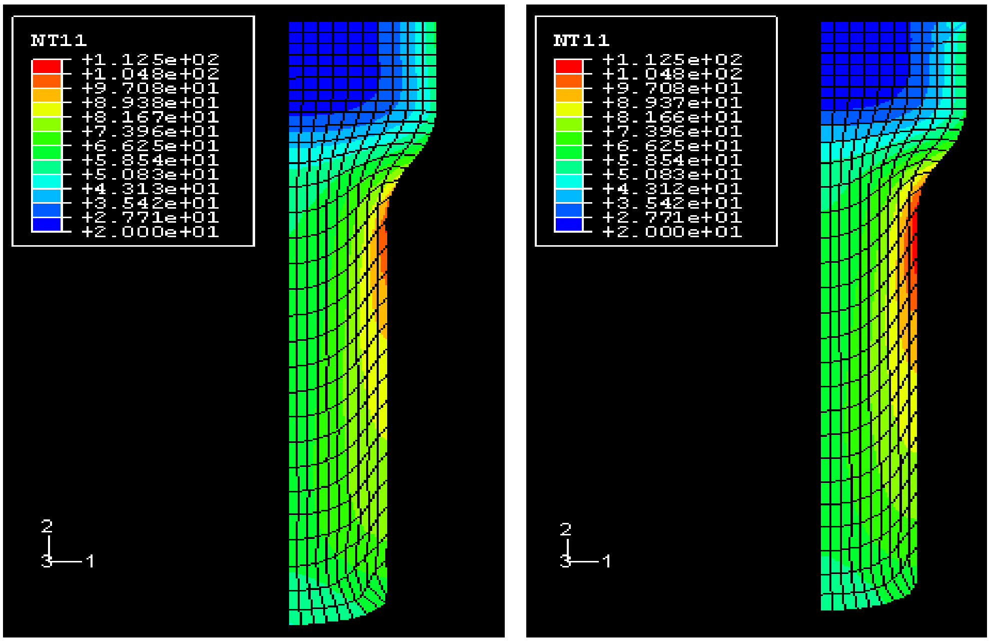


Figure 5.5.Temperature contours: Step 2, ABAQUS/Standard using surface-to-surface contact formulation; thermally coupled analysis, left; adiabatic heat generation (without heat generation due to friction), right.

As noted earlier, excellent agreement is observed for the results obtained with ABAQUS/Explicit (using both the default and enhanced hourglass control) and ABAQUS/Standard. Fig.5.6 compares the effects of ALE adaptive meshing on the element quality. The results obtained with ALE adaptive meshing show significantly reduced mesh distortion. The material point in the bar that experiences the largest temperature rise during the course of the simulation is indicated (node 2029 in the model without adapitivity).

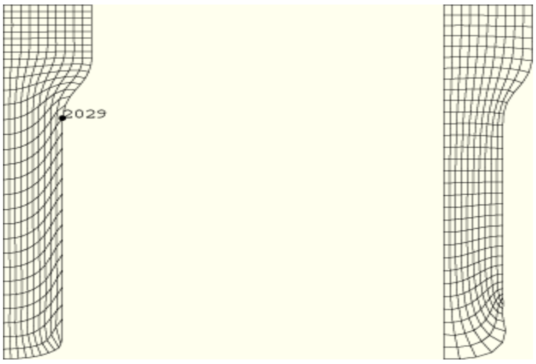


Figure 5.6 Deformed shape of the workpiece: without adaptive remeshing, left; with ALE adaptive remeshing, right.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

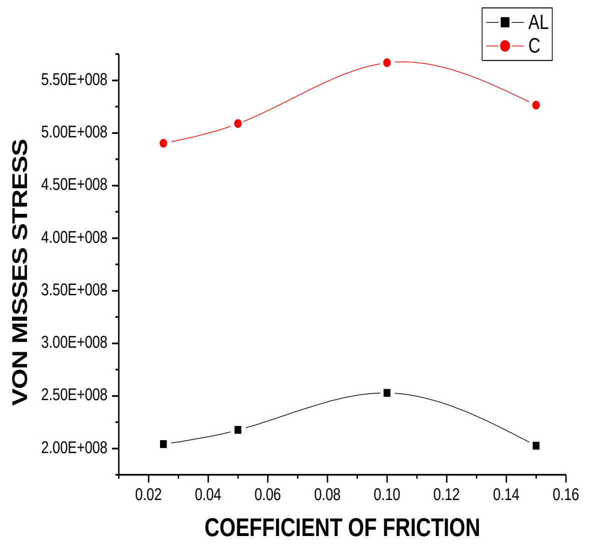


Figure 5.7 Variation of Von-Misses Stress with coefficient of Friction

With the reference to the fig.5.7 it is observed that both Aluminum and Copper as coefficient of friction increases as vonmisses stress increases up to a particular value of coefficient of friction behind this vonmisses stresses shows the decreases in trend. This may be due to excessive heating of material due to higher heat generation at higher values of coefficient of friction.

The vonmisses stresses for copper is much higher as compared to Aluminum. This is due higher modulus of elasticity of Copper as compare to Aluminum.

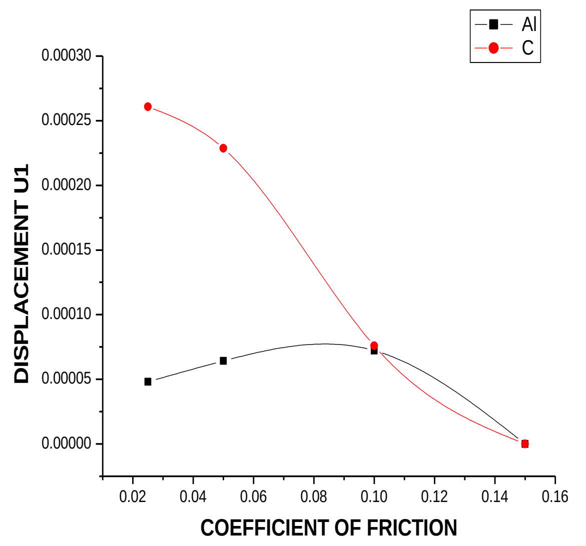


Figure 5.8 Variation of Displacement with Coefficient of Friction

With reference to the fig.5.8. It is observed that the axial displacement decreases as coefficient of friction increases. At a particular value of 0.1 coefficient of friction the axial displacement for Copper & Aluminum are same. Similarly the equivalent plastic strain at 0.1 friction are very close.

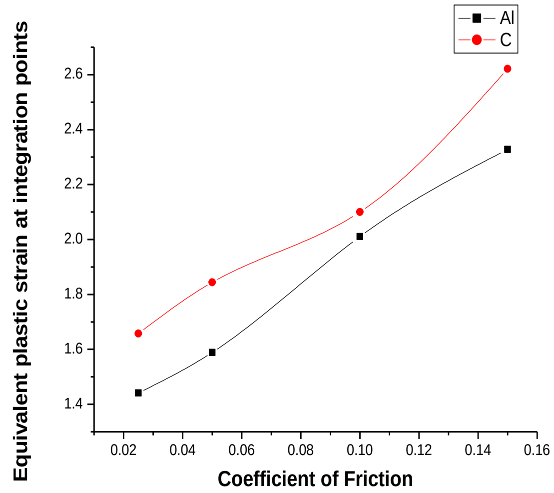


Figure 5.9 Variation of Equivalent Plastic Strain at Integration Points with Coefficient of Friction With reference fig.5.9 the equivalent plastic stain increases for both Aluminum & Copper at 0.10. Coefficient of friction the plastic strains are very closer.

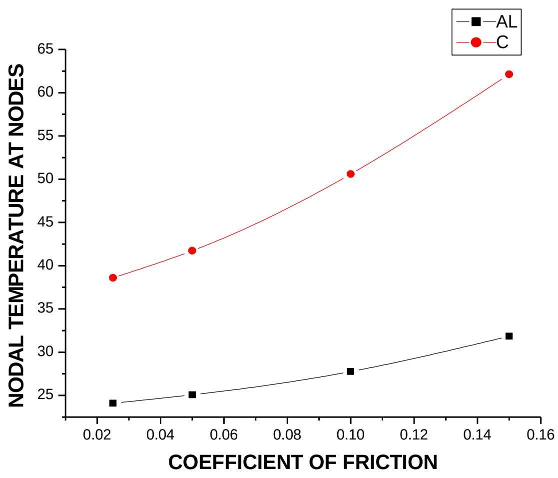


Figure 5.10 Variation of Nodal Temperature at Nodes with Coefficient of Friction

- As temperature increases friction is also increases both for Aluminum & Copper
- As compare to Aluminum friction is more In Copper at higher nodal temperature.
- As a temperature at nodal point decreases the friction comes closer in Copper and Aluminum

VIII. CONCLUSION

- From the contour plots of plastic strain for the fully coupled analysis using CAX4RT elements it is observed that the plastic deformation is most severe near the surface of the work piece, where plastic strains exceed 100%.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

- The contour plot of Von-Misses stress indicate that the peak stresses occur in the region where the diameter of the work piece narrows down due to deformation and also along the contact surface.
- The nodal temperatures obtained using the surface-to-surface contact formulation in ABAQUSs/Standard indicates that the peak temperature occurs at the surface of the work piece because of plastic deformation and frictional heating.
- The peak temperature occurs immediately after the radial reduction zone of the die. This is expected for two reasons.
- First, the material that is heated by dissipative processes in the reduction zone will cool by conduction as the material progresses through the post reduction zone.
- Second, frictional heating is largest in the reduction zone because of the larger values of shear stress in that zone
- By comparison of both the materials aluminum and copper.
- it is observed that both Aluminum and Copper as coefficient of friction increases as vonmisses stress increases up to a particular value of coefficient of friction behind this vonmisses stresses shows the decreases in trend.
- The axial displacement decreases as coefficient of friction increases.
- The equivalent plastic stain increases for both Aluminum &Copper at 0.10.coefficient of friction the plastic strains are very closer.
- As compare to Aluminum friction is more In Copper at higher nodal temperature.

REFERENCES

1. Fuh-Kuo Chena, Wen-Chan Chuang, Shan Torng "Finite element analysis of multi-hole extrusion of aluminum-alloy tubes" Journal of Materials Processing Technology 201, Volume 150–155, 2008.
2. Lontos.A.E, Soukatzidi.F.A, Demosthenou.D.A, Baldoukas.D.K, "Effect Of Extrusion Parameters And Die Geometry On The Produced Billet Quality Using Finite Element Method" Greece Manufacturing Engineering (ICMEN), Volume 1-3, 2008.
3. Tiernan.P, Hillery.P.T, Draganescu.M and Gheorghe.M, "Modelling of cold extrusion with experimental verification" Journal of Materials Processing Technology,Volume 168, Issue 360–366, 2008.
4. Flitta.I and Sheppard.T, "Nature of friction in extrusion process and its effect on material fiow" Volume 139, Issues 331972-344969
5. Alexander.J.M, "Hydrostaticextrusion". ISBN10: **0263517098 / 0-263-51709-8** ISBN13: **9780263517095,** Publication Date: **1971.**
6. Betzalel Avitzur, "Metal forming processing analysis, McGraw-Hill education, ISBN:007002510X, EAN:9780070025103
7. Bridge man.P.W, "Physics of high pressure". Dover Publications, 1970 - 398 pages
8. Carmai.S.J.J, Pitakthapanaphong.S, Sechjarern.S, "3DFinite element analysis of metal flow in hot aluminium extrusion of T-shaped profile with various offset pockets". Volume 31, ISSUE 2, 2008.
9. Drozda et.al., "Tool and Manufacturing Engineers Handbook: Forming", Volume 2, 1984, SME, ISBN 0872631354.
10. Fuchs, "Hydrostatic pressure and its role in metal forming",Mechanical Engineering. Conover Mast Publications., 1967
11. Gosh.A and Mallik, "Manufacturing Science". Ellis Horwood, 1986-433 pages
12. Lambert and other "A theory on the mechanical of axisysmmetric exit through conical die". Volume 91, Issues 10696-24063, 1988.
13. Oberg et.al., "Machinery's Handbook", New York: Industrial Press, ISBN 0-8311-2635- Volume 3, 2008.
14. Pugh "Redundant work and friction in the Hydrostatic Extrusion",Mech.Engg. Science,1964.
15. Sortais.H.C and kobtyshi, "An optimum die profile for axisymmertric extrusion." Publisher Society of Manufacturing Engineers, 2003.
16. Yenger and Rice "Fluid flow Lubricating in the Hydrostatic Extrusion" Mechanical Engineering. Publisher Mc Graw-Hill book company, inc., 1935.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)