EFFECT OF DIE SHAPE ON THE TEMPERATURE AND STRESSES DISTRIBUTION IN THE COMPOUND (FORWARD-BACKWARD) EXTRUSION PROCESS

Rana Ali Hussien
Pumps Engineering Department
Technical College /Al-Musaib – Iraq.

ABSTRACT
In this research the effect of the die geometry in the compound extrusion (forward – backward) process on the distribution of temperature and effective stresses is investigated; Effect of the geometrical shape of extrusion process on the power required for the forming process was also investigated. The flat surface for the punch is chosen and two types of die shape (flat and arc) are studied. The analysis of temperature and stresses distribution in the die and billet is studied using finite element method via ANSYS software. The contact region was taken into consideration. From the results, it was noted that there a tiny changes in the pattern of temperature distribution and also a huge changes in the equivalent stress levels due to the changing in die geometry. In the backward extrusion process, a significant effect on the stress levels was also noticed due to changing in die geometry. At last, the results show that the best results of was achieved at arc die geometry .

KEYWORDS: Extrusion , Finite Element Method, ANSYS ,Temperature distribution, Stress analysis, Forward-Backward Extrusion, Compound Extrusion.

تأثير الشكل الهندسي للقالب على توزيع درجات الحرارة والاجهادات المكافئة في عملية البثق المركب (الخلفي – الامامي)

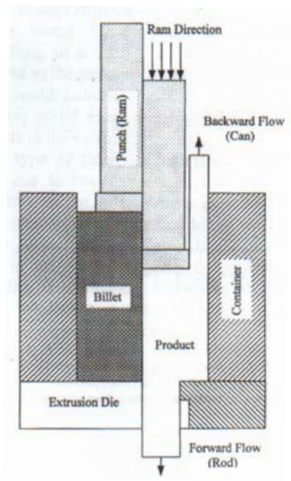
رنا علي حسين
الكلية التقنية / المسيب – قسم هندسة المضخات

الخلاصة
في هذا البحث تم دراسة تأثير الشكل الهندسي للقالب في عمليه البثق المركب (الخلفي –الامامي) على توزيع درجات الحرارة والاجهادات المكافئة وللتحقيق من تأثير عوامل الشكل الهندسي لعملية البثق المركب على كل من متطلبات القدرة اللازمة للتشكيل . تم اختيار شكل هندسي مسطح لرأس المكبس (المخرم) وبنوعين للشكل الهندسي للقالب (مسطح و قوس). حيث تم تحليل توزيع درجات الحرارة على القالب والمعدن ودراسة الاجهادات المكافئة وبواسطة طريقة العناصر المحددة من خلال استعمال برنامج الـ ANSYS حيث تم أخذ منطقة التلامس بين الاسطح المتلامسة بنظر الاعتبار وقد بينت النتائج بانه يحصل تغيير طفيف في شكل توزيع درجات الحرارة نتيجة تغيير شكل القالب (مسطح و قوس) ولكن يحدث تغيير ملحوظ في الاجهادات المكافئة وايضا هناك تغيير واضح للاجهادات في البثق الخلفي نتيجة تغيير شكل القالب وان افضل النتائج كانت عندما يكون سطح القالب على شكل قوس.

INTRODUCTION

Despite the increasing demand for studying the effects of die profile or pass geometry in different extrusion processes, so far very little systematic work has been carried out in combined backward –forward extrusion process for prediction of detailed metal presentation and assessment of redundant strain levels, load requirements as a relative pressure,which should be taken into consideration in order to design the extrusion process and to control the material properties of products effectively. There is a continuing interest in homogeneity of the structure and properties of extrusion products, especially for complex sections. These can often be related to die design, and die profiles can be calculated to reduce redundant wok, improve uniformity of strain, or avoid defects. The finite element method has been applied to analysis this process. Many analytical and numerical methods are investigated like Slab method, Slip line field and Hodographs method, these methods are very complicated and needed many boundary conditions. Johnson et al, 1985, used method to determine the deformations with increasing the strain, this method known “upper bound theorem”. Azel, 2002, used visco- plasticity method to analyze the extrusion process in which the flow of metal was analyzed during deformation in which the stress and strain can be determined. Finite element Method is used during the last decade which is the powerful method to solve the forming metals problems. Nayak and Zienkiewicz 1972, demonstrated the way of analyzing elasto-plastic problems by FEM. Abmijit and Subrata 1984 and Owen and Hinton 1982, are used the FEM to solve the forming problems through visco-plasticity analysis. This method is a general method and can be detect the flow of metal, temperature, stress and deformation contours in case of linear and non linear materials. Chandra and Rama 1992 used the FEM to study the plastic deformation in solid, superplasticity and visco-plastic materials. Hani, 1998 used the FEM in three dimensional domain to analyze the coupled heat-structure field. LLi and Duszczyk 2003 studied a three dimensional FEM simulation on the isothermal extrusion of a 7075 aluminum billet with a predetermined non- linear temperature distribution. Ayad, 2005 is investigated the pass geometry in compound backward-forward extrusion process. Several packages are existing like ALPID, ABAQUS, ANSYS …etc to analyze the forming of metals. ANSYS software is used in this research to demonstrate the extrusion process and to show the temperature distribution of flow of metal and its equivalent stresses. During the surveying of the previous researches, it can be notice that it is studied the flow of metal without effect of temperature. In our research, the temperature distribution and the equivalent stresses during the compound extrusion process is investigated it is the novel analysis of the forward-backward extrusion by ANSYS software. (Fig.1 shows the compound extrusion) . A complete model is built by ANSYS language (Ansys Parametric Design Language (APDL)) to analyze the forward-backward extrusion process.

Fig.1 Compound extrusion process (Ayad, 2005)



MATERIAL PROPERTIES

The material of the billet is aluminum alloy (Al- 0.53Mg- 0.7Si) with chemical composition

Element	Si	Cu	Fe	Mg	Mn	Zn	Ni	Rem
%	0.7	0.14	1.35	0.53	0.04	0.03	0.04	96.75

and the mechanical properties are, the yield stress , $\sigma y = 150$ KPa , Young Modulus of elasticity E=72 MPa , Possion's ratio $\upsilon = 0.4$.

MODEL GENERATION

The ultimate purpose of a finite element analysis is to re-create mathematically the behavior of an actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype. In the broadest sense, the model comprises all the nodes, elements, material properties, real constants, boundary conditions and the other features that used to represent the physical system.

In ANSYS terminology, the term model generation usually takes on the narrower meaning of generating the nodes and elements that represent the special volume and connectivity of the actual system. Thus, model generation in this study will mean the process of defining the geometric configuration of the nodes and elements.

Drawing the two dimensional specimen model and meshing using meshtool. Solid modeling is usually more powerful and versatile than other modeling, and is commonly the preferred method for generation models. The two Dimensional model of specimen is done by plotting and meshing two dimensional axisymmetric plane with elements. The billet element is Plane13 (coupled field- element) and the contact condition is used via elements contac48 (Fig.2 shows the axisymmetric model) .

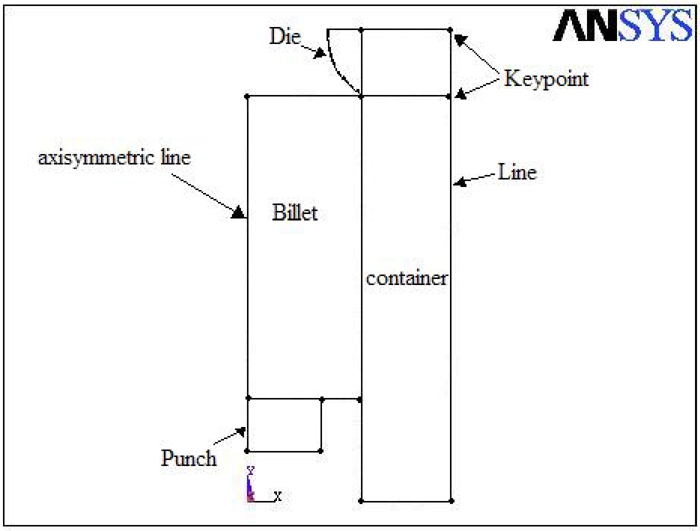


Fig.2 Axisymmetric model

ANSYS software is used to analyze many cases in linear and non-linear (Saeed Moveni, 1999). In this study ANSYS software is used to analyze, study the compound extrusion process, temperature and stresses distribution in the billet and the die with different die shapes (flat and arc). The model was generated for both geometrical and material nonlinearity and coupled field in addition to contact case between the die, billet and punch (ANSYS,1999).

Due to symmetry, half of the punch, die and billet are modeled. The solid modeling is used in ANSYS to make the mesh with element type plane13 for the die , punch and billet and take into consideration the contact region which is meshed with element type contac48. Fig.3 shows the meshed model and Fig.4 shows the flowchart for the numerical simulation is shown below :

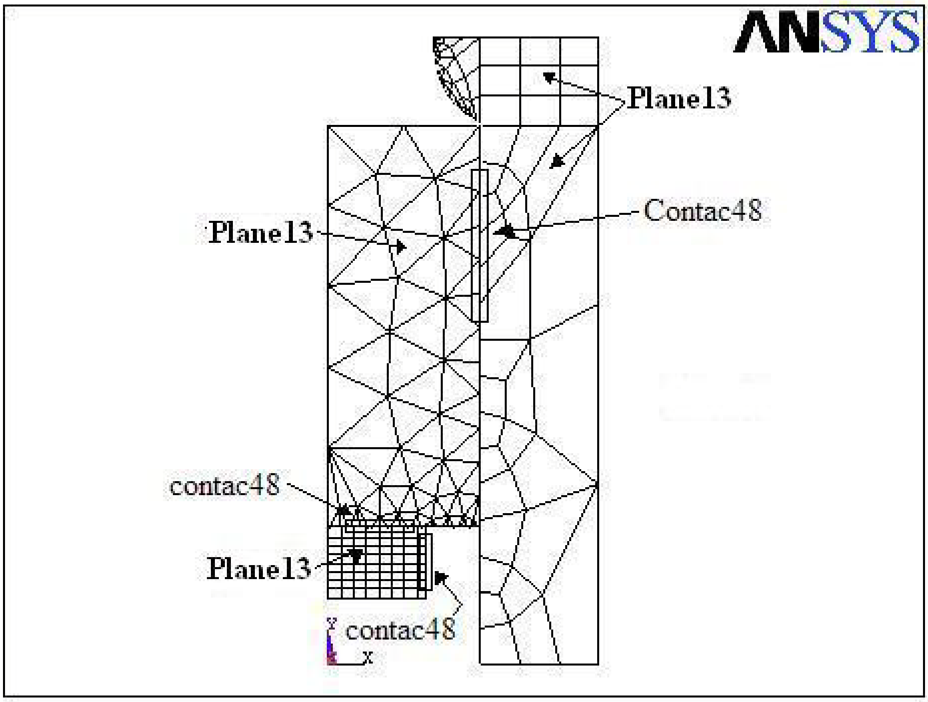


Fig.3 Mesh the model of compound extrusion

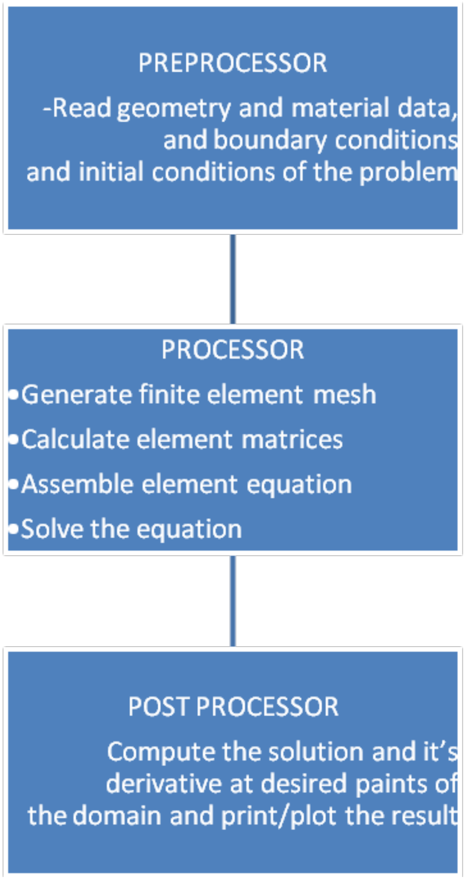


Fig.4 flowchart for the numerical simulation

COUPLED – FIELD MODEL

A coupled – field analysis is an analysis that takes into account the coupling between two or more fields of engineering. In this research, the interaction between the structure and thermal effect is studied .

There are basically two methods of coupling distinguished by finite element formulation techniques used to develop the matrix equations. They are strong coupling and weak coupling. Thermal – Structural Analysis in ANSYS software is built on the strong coupling via element Plane13 with DOF; UX,UY,TEMP, and the matrix equation is :

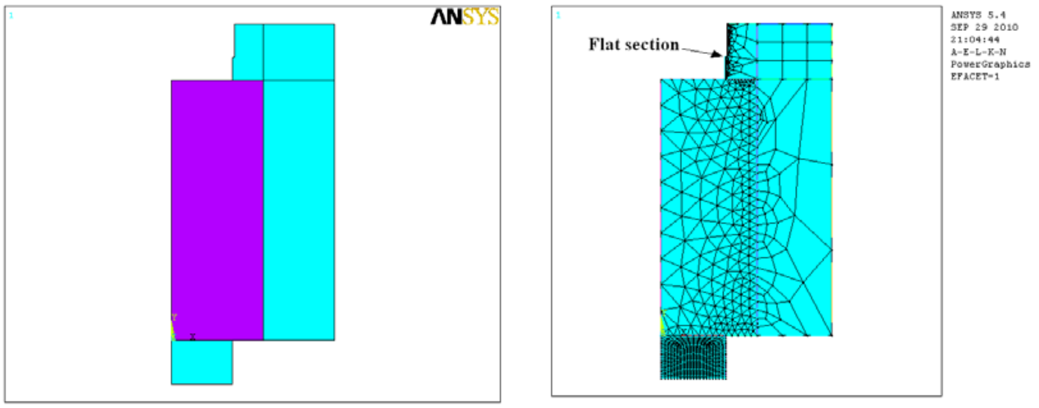
$$\begin{bmatrix}[M] & [0]\\ [0] & [0]\end{bmatrix}\begin{Bmatrix}\{\ddot{u}\}\\ \{\ddot{T}\}\end{Bmatrix} + \begin{bmatrix}[C] & [0]\\ [0] & [C^t]\end{bmatrix}\begin{Bmatrix}\{\dot{u}\}\\ \{\dot{T}\}\end{Bmatrix} + \begin{bmatrix}[K] & [0]\\ [0] & [K^t]\end{bmatrix}\begin{Bmatrix}\{u\}\\ \{T\}\end{Bmatrix} = \begin{Bmatrix}\{F\}\\ \{Q\}\end{Bmatrix}$$

In this research a complete APDL program is built to solve the transient coupled field equation of forward-backward extrusion as shown in Appendix-A- .

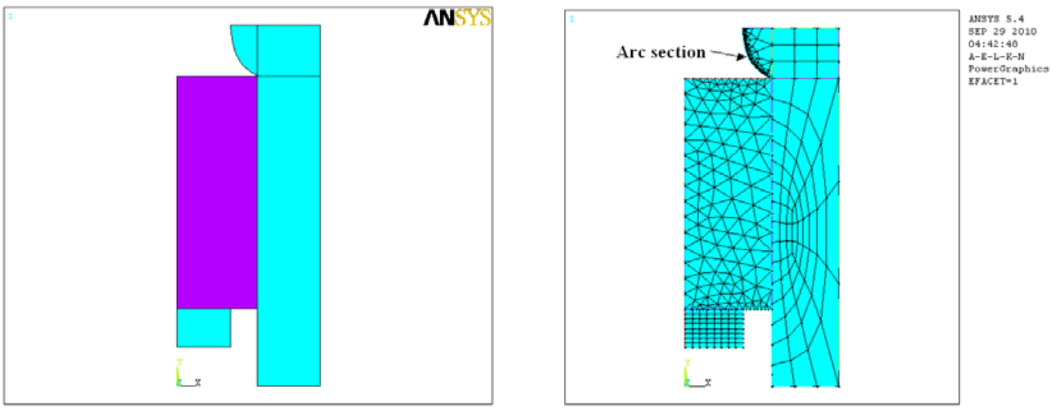
Where [M] .. mass matrix, [C] .. damping matrix, [K]..the stiffness matrix (structure), {F} force vector, $[K^t]$.. stiffness matrix (thermal), {u}.. displacement vector and {T} .. Temperature vector.

CASES STUDIES

Two cases are investigated. The first one is for flat die and the another for arc die as shown in Fig.5.



Case one- Flat Die



Case two - Arc Die

Fig.5 Flat and arc die model for forward-backward extrusion

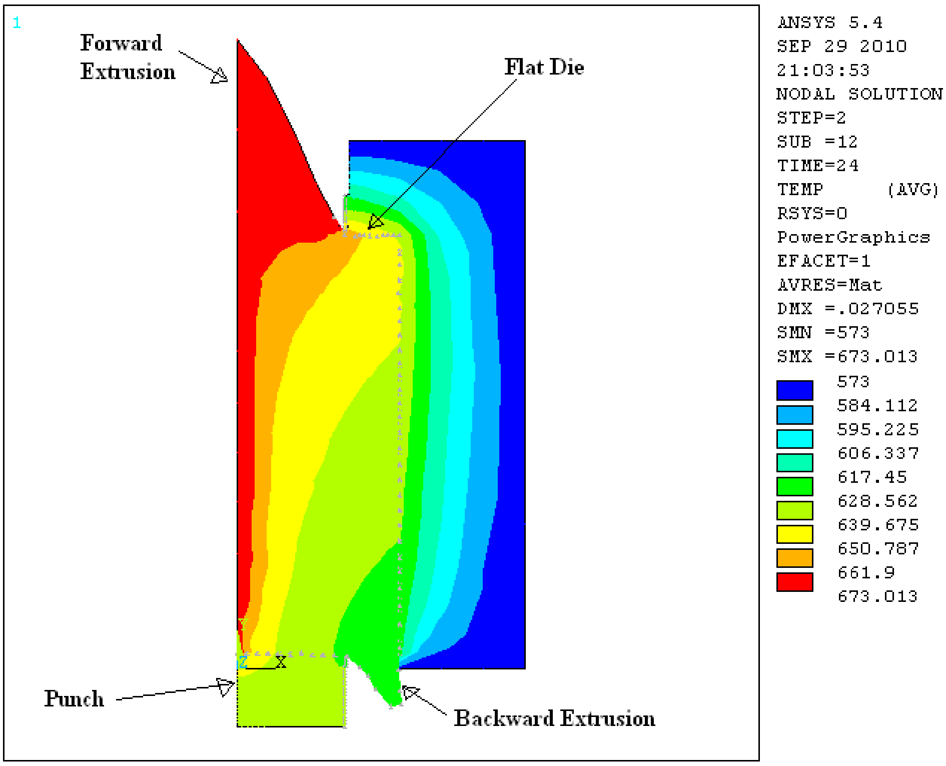
RESULTS AND DISCUSSION

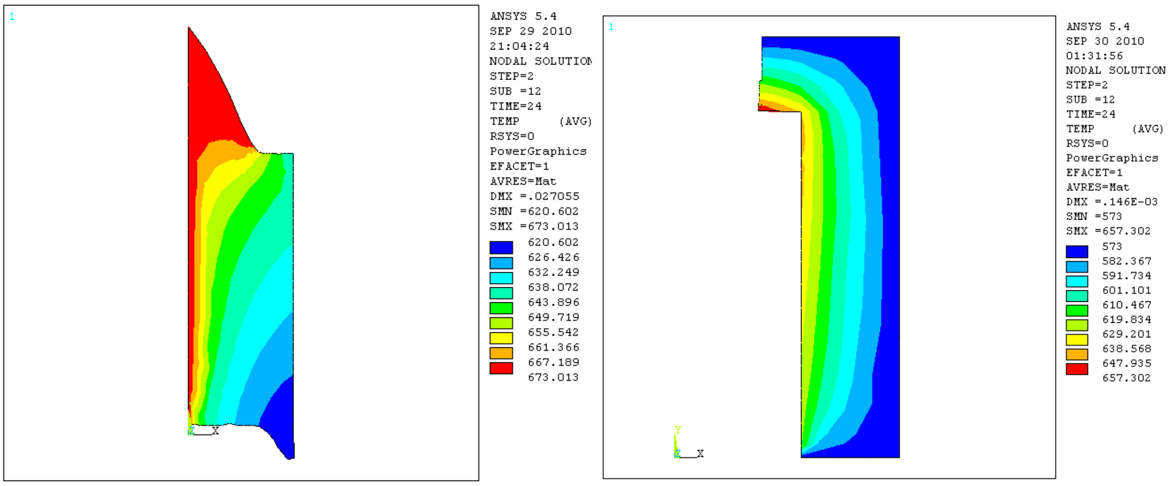
Two cases are studied to investigate the effect of die shape on the temperature and stresses distribution for forward-backward extrusion process by finite element method via ANSYS software. The first case is the extrusion process with flat die. Fig.6 shows the temperature distribution with flat die; and Fig.7 shows the equivalent stresses in with flat die. Fig.8 shows the temperature distribution with arc die; and Fig.9 shows the equivalent stresses with arc die.

More reliable compound extrusion products may be achieved by giving more attention to the punch heat and die surface design. Because of the complex deformation situation during the compound extrusion and availability of more than one type of velocity field through the process duration, ANSYS analysis may be the superior for application.

The complexity of flow situation during the combined backward- forward extrusion process makes it preferable to be analyzed by ANSYS software rather than other methods like the upper bound analysis method . This is because of the difficulty of selecting a general type of velocity field for analyzing this process, especially when there are forward-backward extrusion and different die surface profiles.

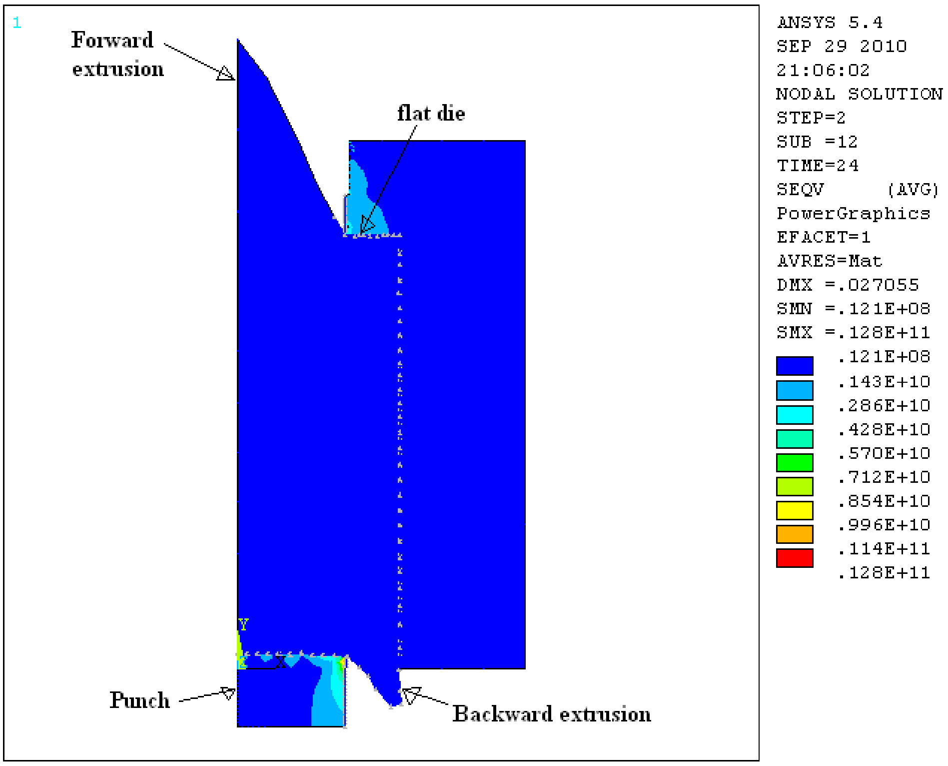
For illustration of the metal flow behavior, the photos of different cases of combined backward-forward extrusion are included in the Figs.6,7,8 and 9. All cases explain how the quantity of metal flow could be affected with the restriction to the metal flow in the forward direction. This restriction to the metal flow is affect by the combination of different die profile.

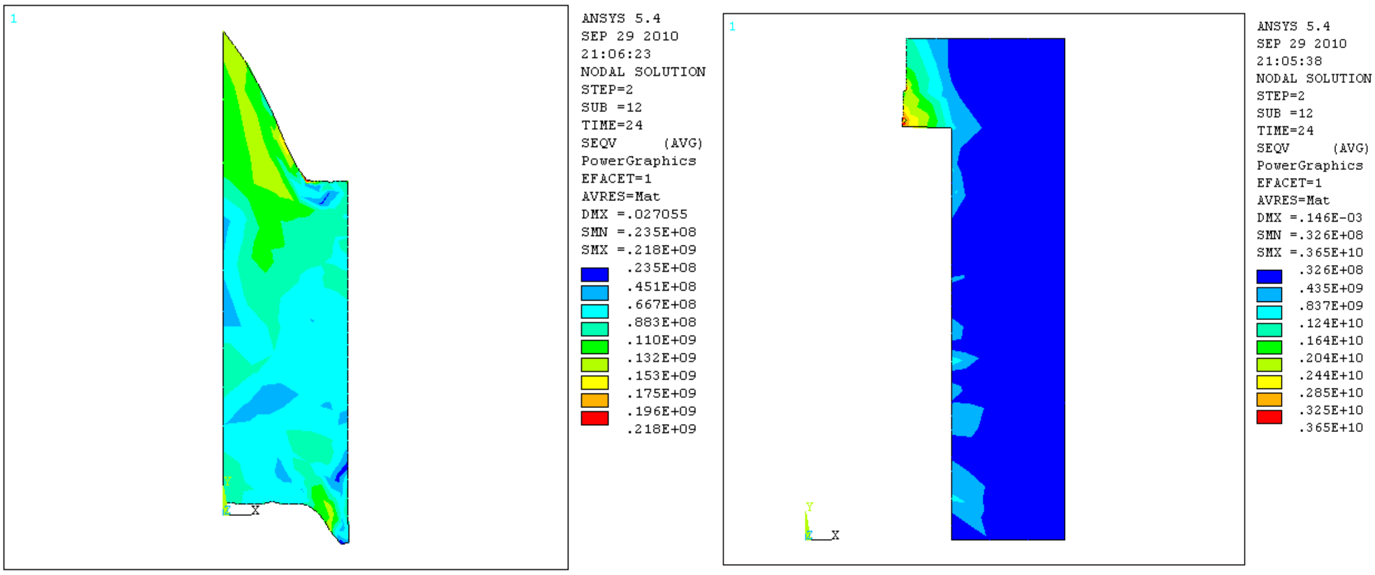




Billet Die and container

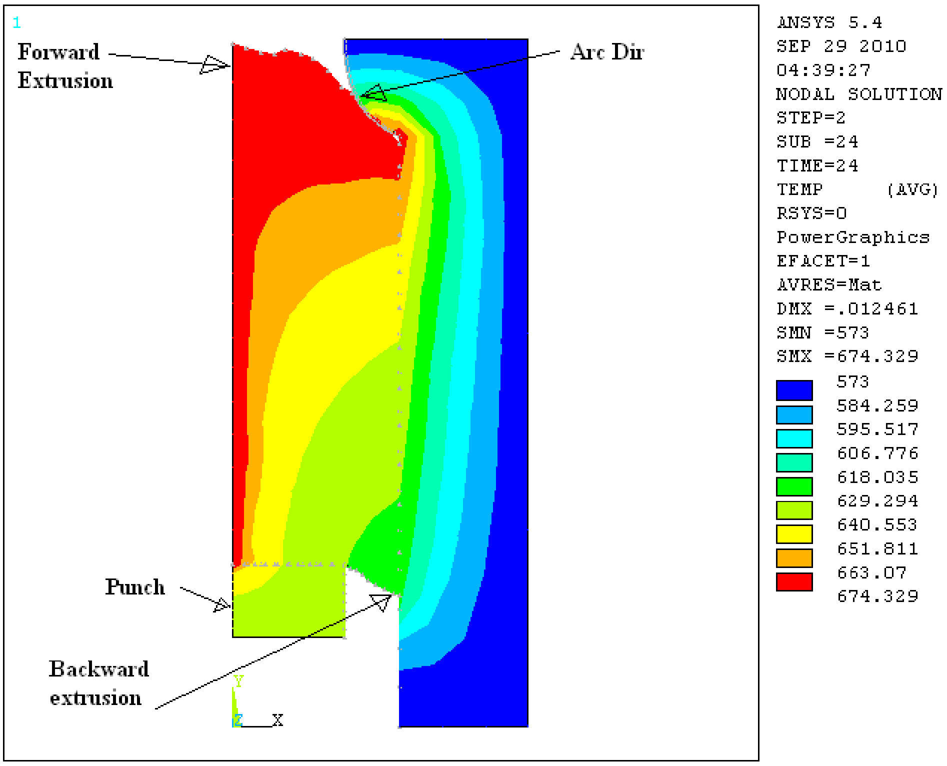
Fig.6 Temperature distribution (°C) in the billet and the flat die

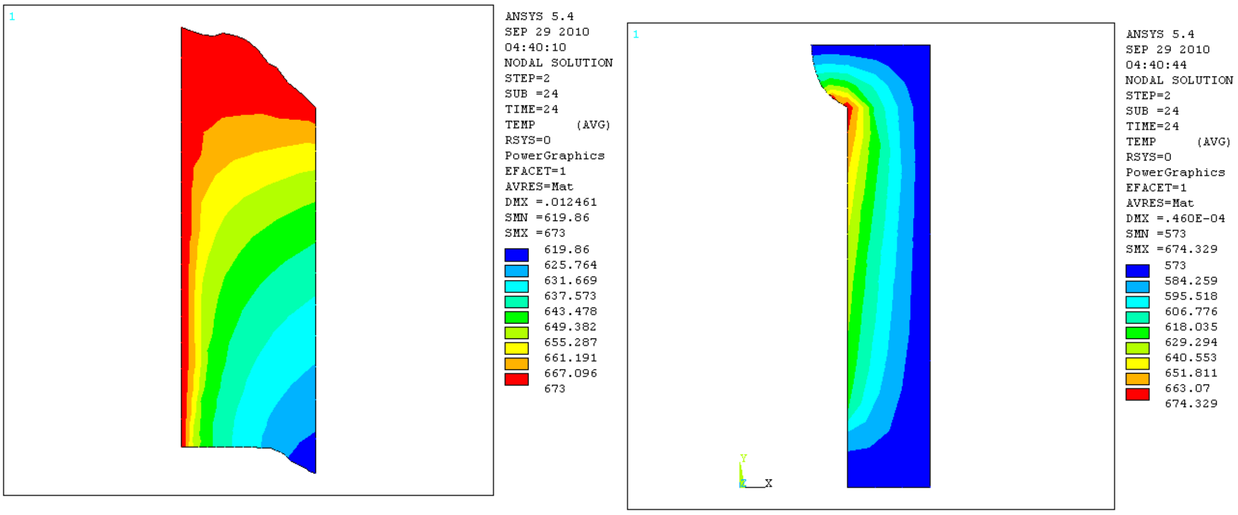




Billet Die and container

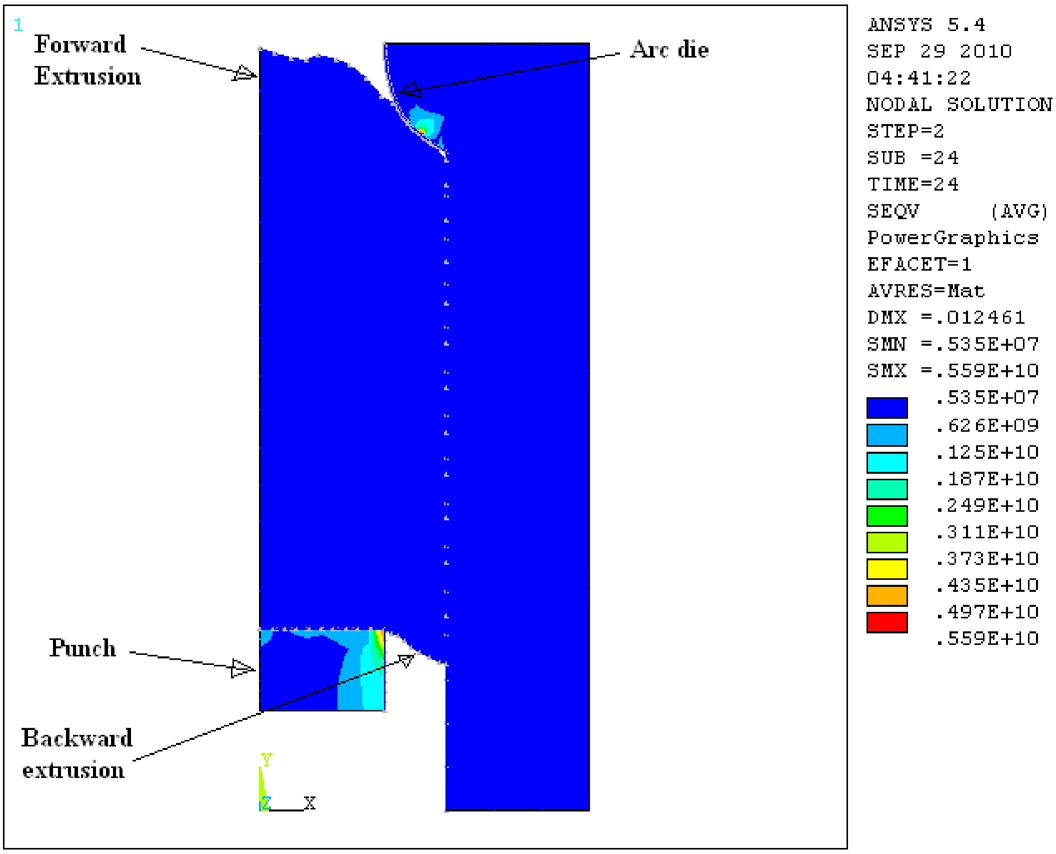
Fig.7 Equivalent stresses distribution (N/m^2) in the billet and the flat die

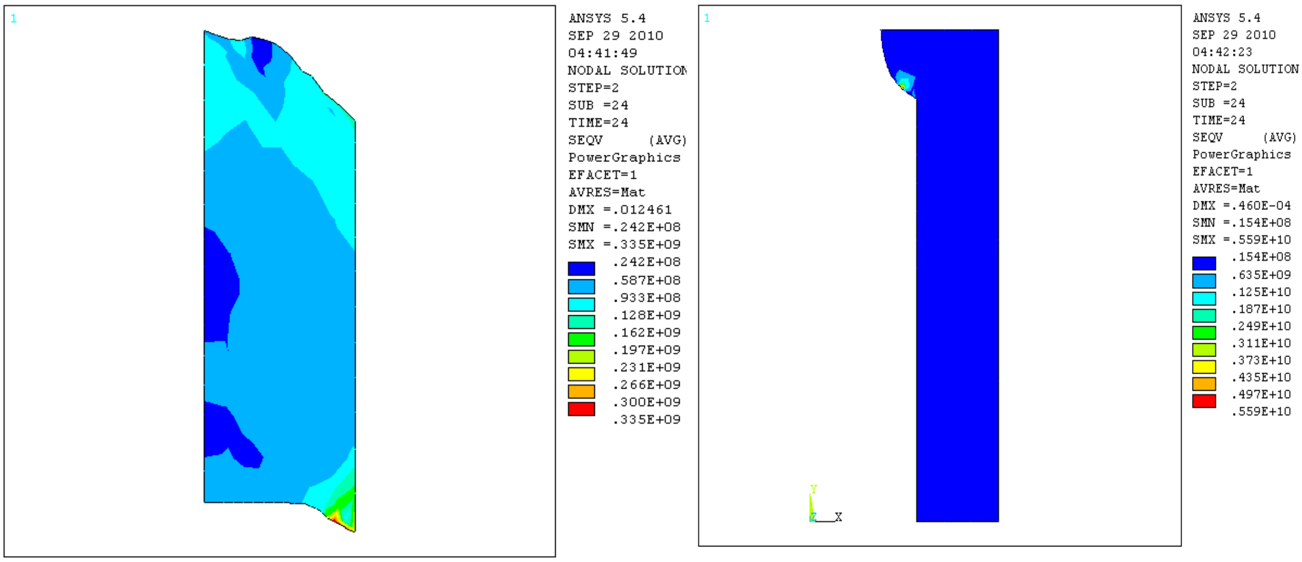




Billet Die and container

Fig.8 Temperature distribution (°C) in the billet and the arc die





Billet Die and container

Fig.9 Equivalent stresses distribution (N/m^2) in the billet and the arc die

CONCLUSIONS

For the given combinations of pass geometry for compound backward – forward extrusion process , which have been investigated analytically by ANSYS, the following points had been concluded:

1- There is a significant effect of the backward extrusion when they are combined with the forward.
2- The streamline profiles of the die surface have an important effect in homogeneity of the material flow in the forward direction.
3- Because of the complex deformation situation during the compound extrusion and availability of applying the velocity field through the process duration, ANSYS analysis may be the superior for application.
4- The availability of using ANSYS program to make the extrusion process and to show the flow of metal, temperature and stresses distribution during extrusion process.
5- Simple change in the distribution of the temperature due to die shape (flat – arc) but a huge change in the equivalent stresses.
6- By changing the die geometry, there is a significant effect on the stresses in the backward with little effect on the temperature distribution.
7- The best results may be achieved by arc die.

REFERENCES

[1] Johnson W. and Mellor P.B. “Engineering Plasticity”, 1985.
[2] Azel, R. I. “Numerical simulation of 3-D regular polygonal sections in forwards extrusion by elastic-plastic finite method”, Ph.D. thesis , U.O.T.,Baghdad, Iraq, 2002.
[3] Nayak A.C. and Zienkiewicz O.C. “ Elasto-Plastic stress analysis. A generalization for various constitutive relations including strain softening” . Int. J. Num. Meth. Eng.,No. 5, PP,113-135, 1972.

[4] Owen D.R.J. and Hinton E. “Finite elements in plasticity”, Pinerdge press limited, Swansea, U.K., 1982.
[5] Abmijit,C. and Subrata, M. “A finite element analysis of metal forming problems with an elastic – viscoplastic material model” Int. J. Num. Meth., vol.20, 1613-1628, 1984.
[6] Chandra , N. and Rama, S.C. “ Application of finite element method to the design of superplastic forming process” Trans. ASME, vol. 114 , Nov., 1992.
[7] Hani Aziz Ameen, “Finite element Method for thermo Elasto-Plastic Stress Analysis of Three Dimensional Problems”, Ph.D. thesis, University of Technology, Baghdad- Iraq, 1998.
[8] Lli, J. Zhou and J. Duszczyk, " A 3D FEM Simulation study on the isothermal extrusion of a 7075 aluminum billet with a predetermined non- linear temperature distribution" , Modelling and Simulation in Materials Science and Engineering, Vol. 11, No.3 , pp 401-416, 2003.
[9] Ayad F. Shahab " Investigation of the pass geometry in compound backward-forward extrusion process", Ph.D. thesis, University of Technology, Baghdad- Iraq, 2005.
[10] Saeed Moveni, “Finite element analysis , theory and application with ANSYS”, 1999.
[11] ANSYS, “Structural Manual” , Ansys, Inc. (online help), 1999.

APPENDIX – A-
APDL for Forward – Backward Extrusion Process

/prep7	!contact mesh
!plot the die	type,3
k,1,,0	mat,3
k,2,,3/1000	real,3
.	!contact
.	lsel,s,line,,57,58
k,56,,	nsll,s,1
k,57,,0	cm,b1,node
L,1,3	!target
L,3,4	lsel,s,line,,1,22,1
.	nsll,s,1
.	cm,a1,node
rectng,0.0225,0.04,0.023+xx,0.036+xx	nsel,all
Aovlap,all	gcgen,a1,b1
Aglue,all	gcgen,b1,a1
!plot of material	! punch
rectng,0,22.5/1000,0,22.5/1000	rectng,0,15/1000,-10/1000,0
!element of die	type,1
ET,1,Plane13	mat,1
KEYOPT,1,1,4	real,1
KEYOPT,1,3,1	Amesh,5
KEYOPT,1,4,0	finish
!element of material	/SOLU
ET,2,13	ANTYPE,4
KEYOPT,2,1,4	TRNOPT,full

KEYOPT,2,3,1	eqslv,sparse
KEYOPT,2,4,0	NLGEOM,on
!contact element	neqit,299
!==================	nropt,1
ET,3,CONTAC48	outres,all,all
KEYOPT,3,1,1	outpr,all,all
KEYOPT,3,3,1	! Load OF MATERIAL
!mesh of die	lsel,s,line,,67
!===========	nsll,s,1
type,1	D,ALL,Uy,1/1000
mat,1	ALLSEL
real,1	TIME,12
AMESH,3	DELTIM,1
AMESH,4	cnvtol,u,.5,0.4
amesh,5	SAVE
!mesh of material	SOLVE
!=================	SAVE
type,2	.
mat,2	.
Real,2	.
AMESH,1	