International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 3, No. 4, October 2014
© 2014 IJMERR. All Rights Reserved

Research Paper

SIMULATION STUDY OF STUB AXLE FORGING USING DEFORM

Ujjwal Kumar[1]*, Ajay Kumar Mishra[1] and Rajkumar Ohdar[1]

*Corresponding Author: **Ujjwal Kumar** ✉ kashyapujjwal6@gmail.com

Until about mid 60's, the analysis of forging processes was based on analytical methods like slab method, slip line method and upper bound method without the help of computerization. Gradually it shifted to computerization by the arrival of digital computers, thereby accuracy and efficiency of such analysis increased. The Finite Element Numerical methods of analysis of bulk metal forming processes is increasingly applied to analyze forming defects, predict and optimize variables and to predict stress, strain damage in dies and workpiece for preventing premature die-failure & forging defects. These tools are not expert systems and do not have any intelligence built-in. They only give solution for 'what-if' analysis. FEM tools will have to be used by expert designers of processes. FEM will not automatically give the answers to the problems. The Finite Element Numerical methods of analysis can be done by Finite Element Method (FEM) & Finite Difference Method (FDM). Simulation study on DEFORM is based on FEM and provides an approximate but acceptable analysis of Forging process constrains.

Keywords: Simulation, Deform, FEM

INTRODUCTION

Simulation is the process of designing a mathematical or logical model of a real system and then conducting computer based experiments with the models to describe, explaining the behavior of the real system.

The aim is to

- To provide a realistic model of the real forging process
- Determination of variables likes fillet radii, corner radii properly.
- Optimization of flash width, flash thickness.
- Determination of proper billet temperature and die temperature.
- Minimize machining means lowering cost.

[1] Department of Forge Technology, National Institute of Foundry and Forge Technology, Ranchi - 834003, Jharkhand, India

Dimensions	Shrinkage Allowance	Die Wear	Finish Allowance	Final Dimensions
7	0.08	0.95	0.054	8
15	0.08	0.95	0.161	16.2
17	0.08	0.95	0.182	18.2
36.5	0.15	0.95	0.392	38
35.5	0.15	0.95	0.38	37
35	0.15	0.95	0.375	36.5
51	0.15	0.95	0.547	52.7
26	0.15	0.95	0.279	28

HORIZONTAL (X-AXIS) DIMENSIONS (WITH ALLOWANCES)

Total length: 234.6 mm

Calculation of plan area

A (Plan area)

Calculation for volume of the product

V (From plan area)

Calculations: Vertical (y-axis) dimensions (with allowances)

$L_{eq.} = 234.6$

Selection of Material

AISI 1045 STEEL (ρ=7.86gm/mm^3)

WEIGHT = ρv = 2.2kg

Calculation of Billet Size

As the maximum diameter of the product is 103 mm but average diameter remains 45.7 mm, I assumed my billet diameter as 50 mm.

$D_{equivalent}$ = 50mm

L equivalent = ΣL_i = required length

Calculation of Equivalent Length

L_i is calculated by equating the volume of each section according to the law:

$\Pi/4d^2\ L_1 = \pi/4(50)^2\ L_2$

Length of the Stock

ΣL_i = 139.87mm

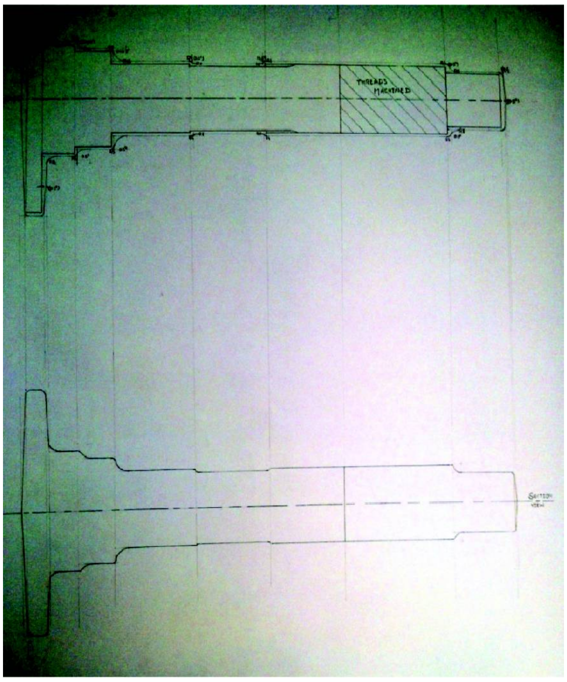
Selection of Draft Angle

For depth/height (H)

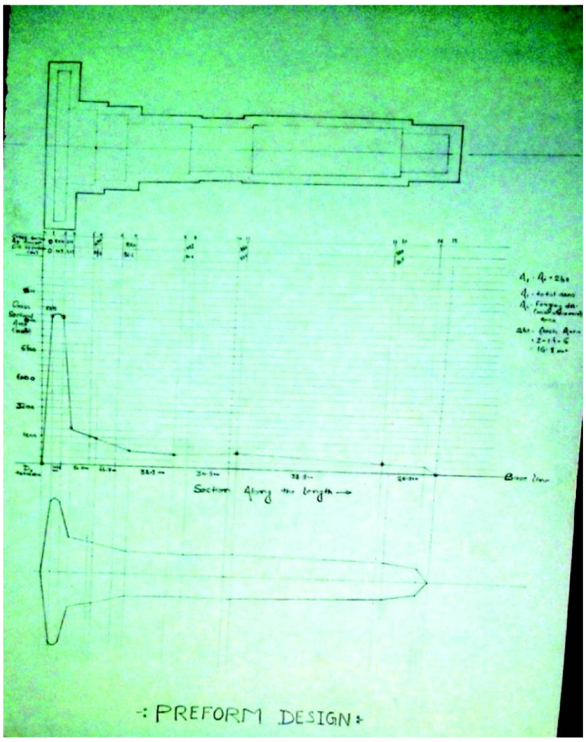
H<12=1^0, H< 12-25=3^0, H>25=5^0

Dimensions	Formulae	Calculation	Value
Flash thickness (t)	T=0.015 √(A)	0.015 √8492	1.38 mm
Flash width(b)	b = 4 x t	b = 4 x 1.38	5.52 mm
Gutter thickness(g)	g = 3 x t	b = 3 x 1.38	4.14 mm
Gutter width(b1)	b1 = 4 x b	4 x 1.14	22.08 mm

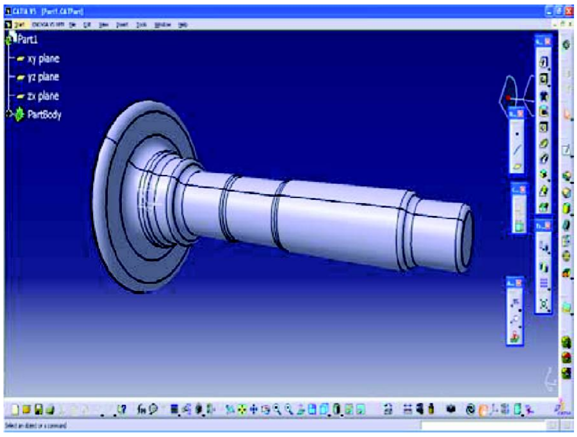
Forging Drawing



Preform Drawing: (Section Line Method)

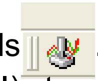


Component Drawing in Catia



Getting Started with Simulation with DEFORM-3D

We will summarize the basic procedure for defining a turning process in DEFORM-3D, and then we will go through each step in detail.

1. Set Simulation controls . Set unit system (English or SI), turn on heat transfer
2. Set object name for workpiece
3. Import workpiece geometry
4. Generate mesh on workpiece
   - Smallest element ½ to 1/5 of feed
   - Size ratio 6 to 8
5. Assign workpiece material
6. Assign workpiece boundary conditions
   - Velocity = 0 on bottom surface
   - Heat exchange with environment on all surfaces
7. Add a second object to the object tree 
8. Import die geometry
9. Generate mesh on die

10. Assign die material
11. Assign die movement
12. Assign boundary conditions
    - Heat exchange with environment on all surfaces
13. Set simulation controls
    - Step -> Solution Steps Definition such that tool moves ¼ element length.
    - Stopping control based on time or distance
14. Object positioning 
    - Rotational position insert to correct angles
    - Mouse drag tool above workpiece
    - Interference position tool down onto workpiece
    - Interference position die sideways into workpiece "shoulder"
    - Offset position "feed" distance *into* workpiece.
    - Interference position insert against end of die.
15. Inter-Object Relationships 
    - Accept default relationships
    - Add relationship: Workpiece Master-Workpiece slave
    - Edit
      – Friction = 0.4 – 0.7
16. Generate contact
17. Check and generate database 

Creating a new Problem File

From the main DEFORM window, click the New Problem icon . Use the DEFORM-

3D preprocessor, and enter a problem name. Follow the setup wizard until the preprocessor opens.

Set Units and Heat Transfer mode

Click the Simulation Controls icon.

 Be sure the unit system is set to English.

Import Workpiece Geometry

Click on **Geometry** and **Import Geo.** Select the file "Workpiece.stl." Check the geometry. It is important the geometry have

- One surface
- No free edges
- No invalid edges

If the geometry has some small errors, the "Fix Geo" option may be helpful. If there are a large number of errors, it may be necessary to repair the CATIA model, and re-export the .stl file.

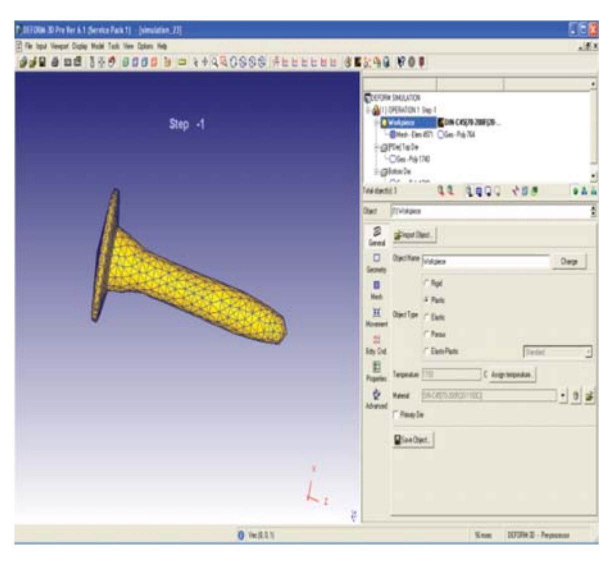
Generate a Mesh on the Workpiece

Click **Mesh.** Go to the **Detailed Settings** tab. set the mesh **Type** to **Absolute.** This means that we will specify element size, rather than number of elements (relative).

Assign Material

Click the **Material** icon. Open the "Steel" folder, and select AISI-1045 Machining.

Click the **Assign Material** button to assign the material to the workpiece.



Assign Boundary Conditions

Click the **Boundary Condition** Icon . Select **Velocity** from the tree. Rotate the workpiece so the bottom is visible, and pick a node on the bottom of the workpiece. Select the direction.

Click the **Add Boundary Condition** icon.

Assign Simulation Controls

Click the **Simulation Controls** icon, and go to the **Step** definition.

Set **Solution Steps Definition** to **With Constant Die Displacement** and assign 0.0015/0.33".

Go to the **Stop** tab, and set the **Primary Die Displacement** in the X, Y, and Z directions.

The simulation will run 305 steps, unless it reaches the stopping criteria first.

Click OK.

Position Objects

Click the **Object Positioning** icon at the top of the interface.

Make dies/workpiece the positioning object.

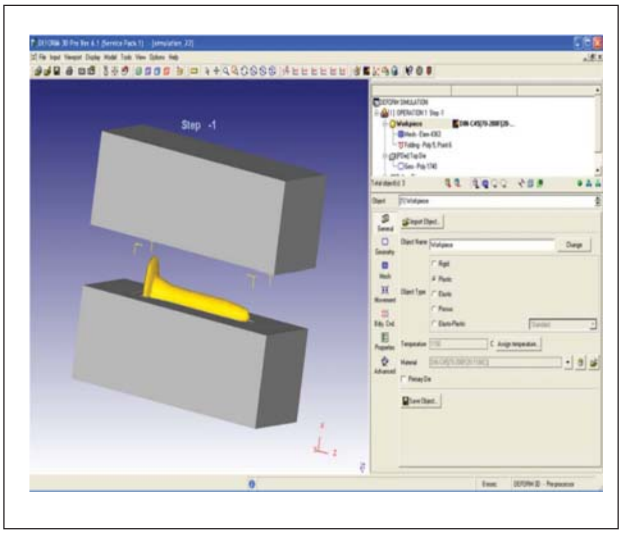
Now use **Interference** positioning to move the dies down until it offset by 100mm from the workpiece.

Be sure the **Positioning object** is the die/workpiece; the **Reference Object** is the lower die.

Make the approach direction –Z, and click **Apply.**

Now we'll position the tool against the shoulder in the workpiece.

The final position should look like the image below.



Click **OK** to get out of object positioning.

Save the data.

Inter-object Relationship

Click the **Inter-Object** icon. The system will offer to assign default relationships.

Accept this.

Click the **Inter-Object** icon, which is right next to the **Positioning** icon.

Click **Edit** to define **friction** and heat transfer values.

Friction modeling is still a matter of some discussion amongst researchers. We have found that, in the absence of better information, values in the range of 0.2-0.4 give reasonable results.

Generating the Database

Click the **Database Generation** icon, next to the **Inter-Object** icon.

Click the **Check** to run the automatic data checking. DEFORM will mark errors with red circles. This indicates a situation which will not allow the situation to run. The user must return to the preprocessor and correct the situation before continuing.

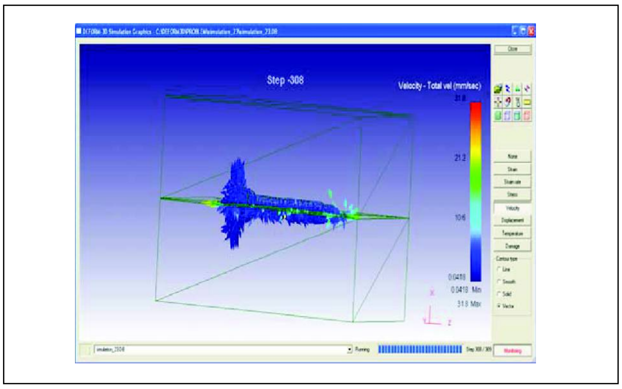
Some conditions will be marked with yellow. These indicate potential problems, which will not necessarily cause a simulation to stop, but may lead to incorrect results.

Simulation Engine

Further after generating the database we close the preprocessor window and move to the next step by submitting the process to simulation engine.

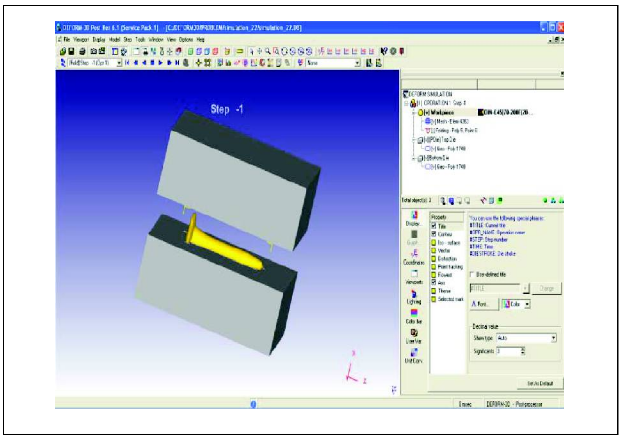
In the section of

we can observe the simulation process

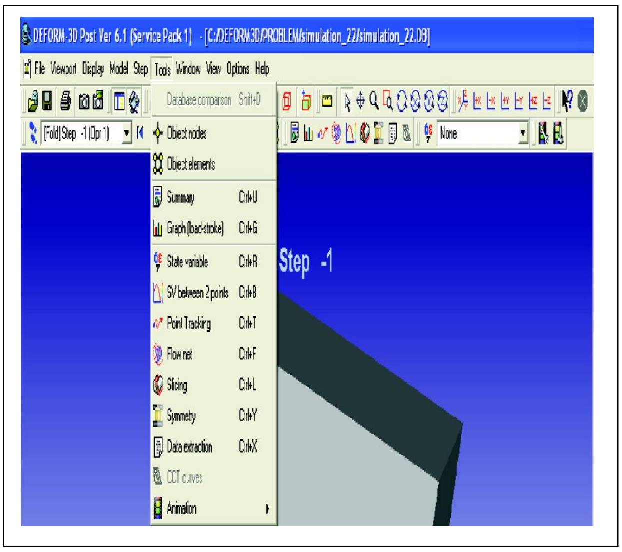


Post Simulation

In post simulation engine the details of process can be studied.



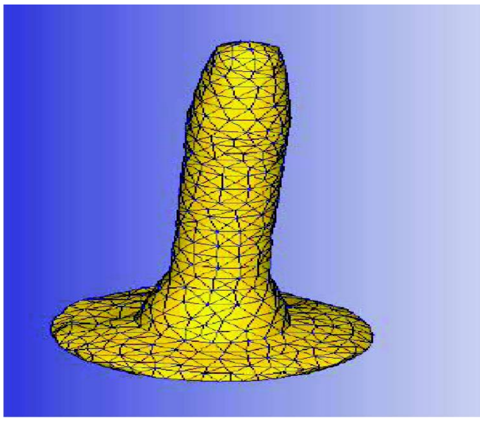
For knowing the different graphs between constraints and check the complete filling we can refer to the section of **DIES:**



Process Variable Considered for Simulation Process

Variables	Value (i=1,2,3)
1. Flash thickness (i)	1.5, 2, 2.5
2. Flash width (i)	5, 5.5, 6
3. Die temperature (i)	200, 300, 400
4. Billet temperature (i)	1050, 1150, 1250
5. Co-efficient of friction(i)	0.2, 0.3, 0.4

Preform



Variable Considered Simulation Wise (Taguchi Experimental Design)

Simulation process chart

Simulation S. No.	Flash thickness	Flash width	Die temperature	Billet temperature	Co-efficient of friction
1.	1.5	5	200	1050	0.2
2.	1.5	5	200	1050	0.3
3.	1.5	5	200	1050	0.4
4.	1.5	5.5	300	1100	0.2
5.	1.5	5.5	300	1100	0.3
6.	1.5	5.5	300	1100	0.4
7.	1.5	6	400	1150	0.2
8.	1.5	6	400	1150	0.3
9.	1.5	6	400	1150	0.4
10.	2	5	300	1150	0.2
11.	2	5	300	1150	0.3
12.	2	5	300	1150	0.4
13.	2	5.5	400	1050	0.2
14.	2	5.5	400	1050	0.3
15.	2	5.5	400	1050	0.4
16.	2	6	200	1100	0.2
17.	2	6	200	1100	0.3
18.	2	6	200	1100	0.4
19.	2.5	5	400	1100	0.2
20.	2.5	5	400	1100	0.3
21.	2.5	5	400	1100	0.4
22.	2.5	5.5	200	1150	0.2
23.	2.5	5.5	200	1150	0.3
24.	2.5	5.5	200	1150	0.4
25.	2.5	6	300	1050	0.2
26.	2.5	6	300	1050	0.3
27.	2.5	6	300	1050	0.4

Source: Experimental variables of simulation

Process Constraints

Flash thickness: 1.5

Flash width: 5.5

Co-efficient of friction: 0.4

Die temperature: 300

Billet temperature: 1100

Complete filling: Yes

Process Constraints

Flash thickness: 1.5

Flash width: 6

Co-efficient of friction: 0.2

Die temperature: 400

Billet temperature: 1150

Complete filling: Yes

Simulation Results: Minitab Worksheet of Analysis Considering Maximum Stress, Maximum Strain, Damage

Minitab - ujjwal project Minitab.MPJ - [Worksheet 2 ***]

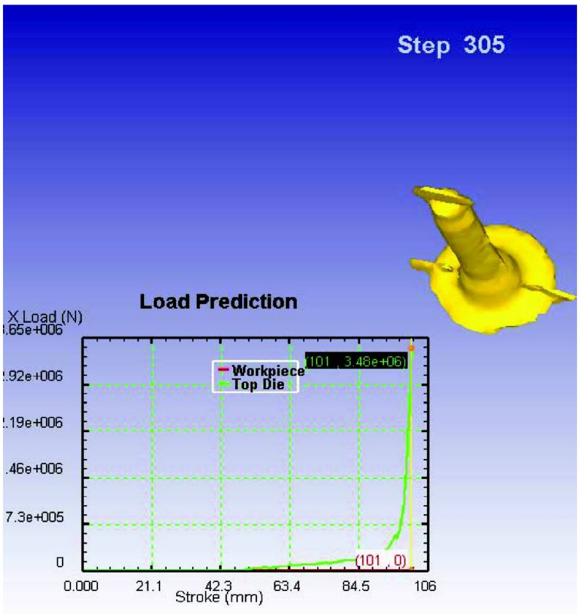
File Edit Data Calc Stat Graph Editor Tools Window Help Assistant

↓	C1	C2	C3	C4	C5	C6	C7	C8	C9-T
	Flash thickness	Flash width	Die temperature	Billet temperature	Co-efficient of friction	Max. Eff. Stress	Max. Eff. Strain	Damage	Filling
1	1	1	1	1	1	59.785	4.5265	0.87257	Incomplete
2	1	1	1	1	2	125.841	3.1988	0.62230	Incomplete
3	1	1	1	1	3	248.746	3.8317	1.00748	Incomplete
4	1	2	2	2	1	192.164	5.4711	1.85683	Incomplete
5	1	2	2	2	2	192.164	5.4711	1.85683	Incomplete
6	1	2	2	2	3	383.999	18.3896	1.04082	Complete
7	1	3	3	3	1	75.803	3.6193	1.57832	Complete
8	1	3	3	3	2	148.819	4.7929	0.89503	Incomplete
9	1	3	3	3	3	144.816	2.6023	0.77142	Incomplete
10	2	1	2	3	1	130.997	2.1995	0.82504	Incomplete
11	2	1	2	3	2	86.507	3.6461	1.30387	Incomplete
12	2	1	2	3	3	115.883	6.9875	1.05034	Incomplete
13	2	2	3	1	1	146.453	5.9842	8.94792	Complete
14	2	2	3	1	2	80.427	4.4749	2.44792	Complete
15	2	2	3	1	3	158.748	3.0013	1.02474	Incomplete
16	2	3	1	2	1	991.411	5.1082	0.85598	Incomplete
17	2	3	1	2	2	80.427	3.5522	0.89031	Complete
18	2	3	1	2	3	136.651	4.4280	1.04083	Complete
19	3	1	3	2	1	128.638	4.4245	0.91069	Complete
20	3	1	3	2	2	124.750	4.4174	2.34406	Complete
21	3	1	3	2	3	102.944	2.4148	0.71637	Incomplete
22	3	2	1	3	1	110.674	4.2021	1.29649	Complete
23	3	2	1	3	2	116.506	5.7952	1.24478	Incomplete
24	3	2	1	3	3	110.674	4.0647	1.07100	Complete
25	3	3	2	1	1	192.999	3.1256	1.09265	Complete
26	3	3	2	1	2	276.180	4.2745	1.40458	Incomplete
27	3	3	2	1	3	210.416	3.5522	0.94440	Complete

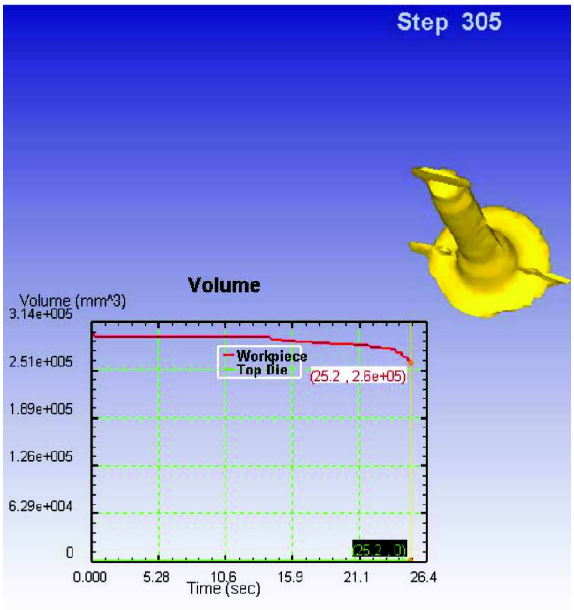
Current Worksheet: Worksheet 2

Simulation 6

Load vs. Stroke Curve

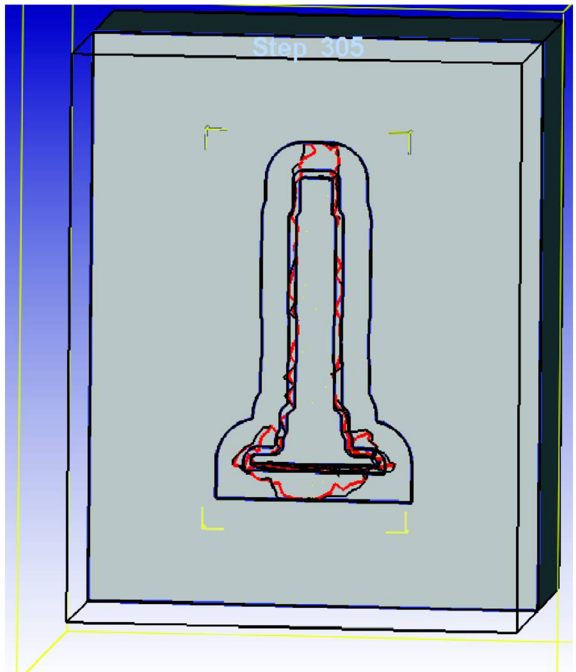


Volume vs. Time



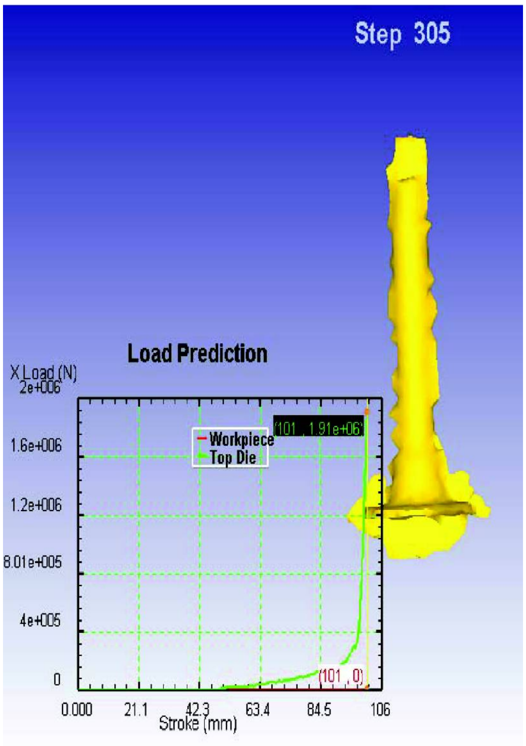
Slicing

Slicing

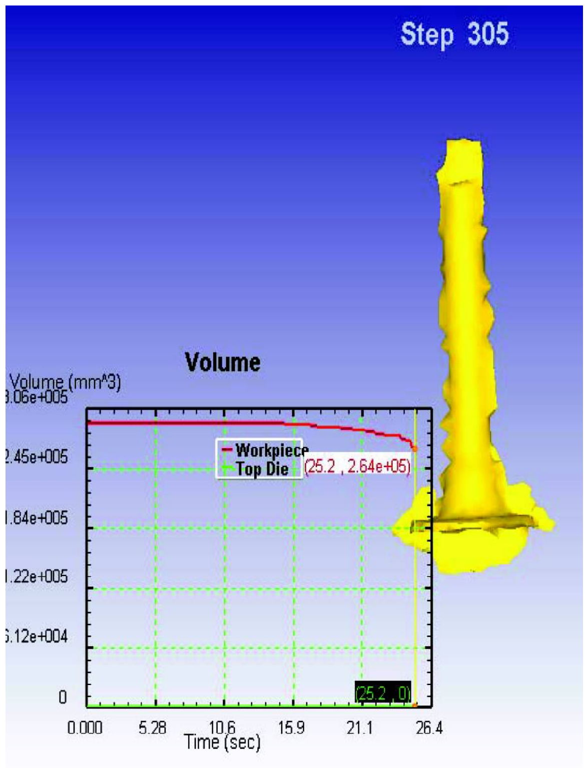


Simulation 7

Load vs. Stroke Curve



Volume vs. Time



Process Constraints

Flash thickness: 2

Flash width: 5.5

Co-efficient of friction: 0.2

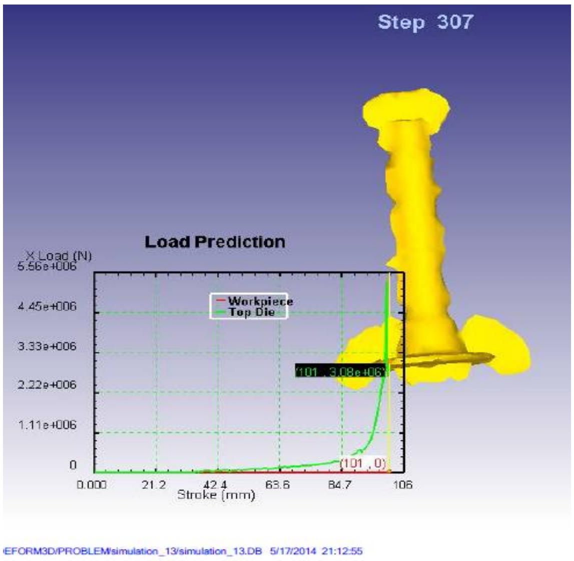
Die temperature: 400

Billet temperature: 1050

Complete filling: Yes

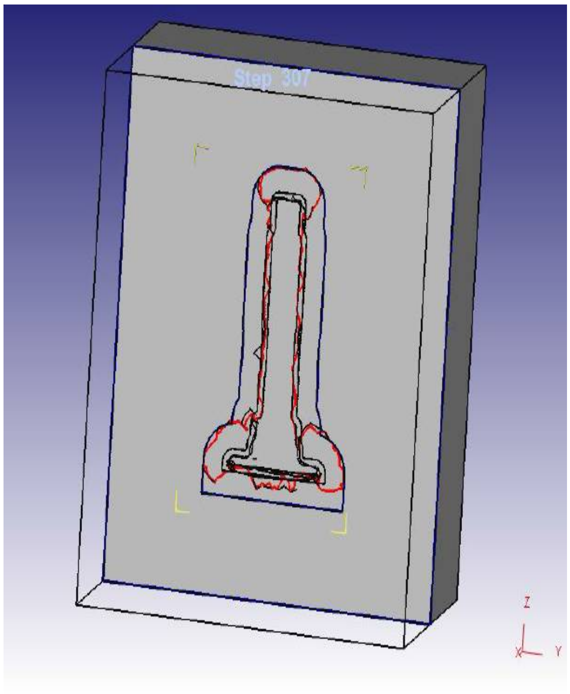
Simulation 13

Load vs. Stroke



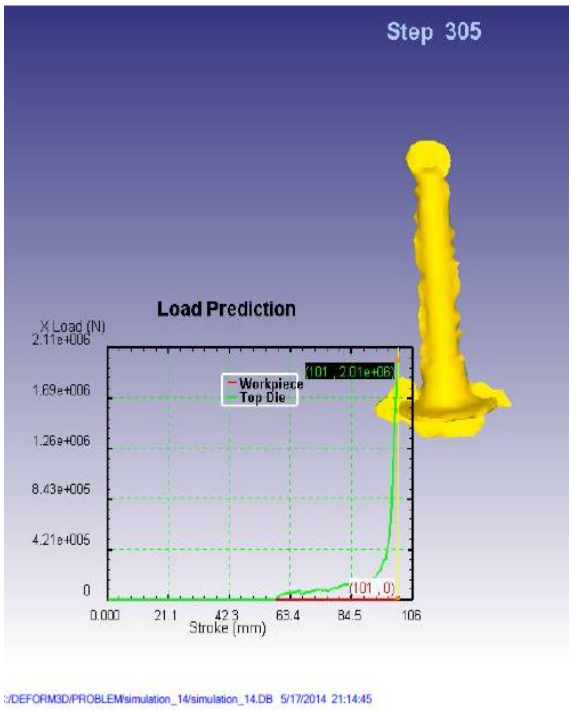
Volume vs. Time

Slicing

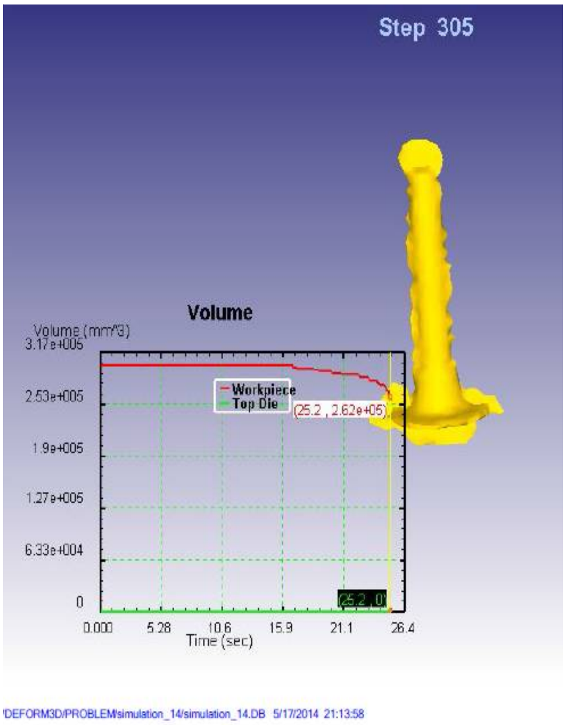


Simulation 14

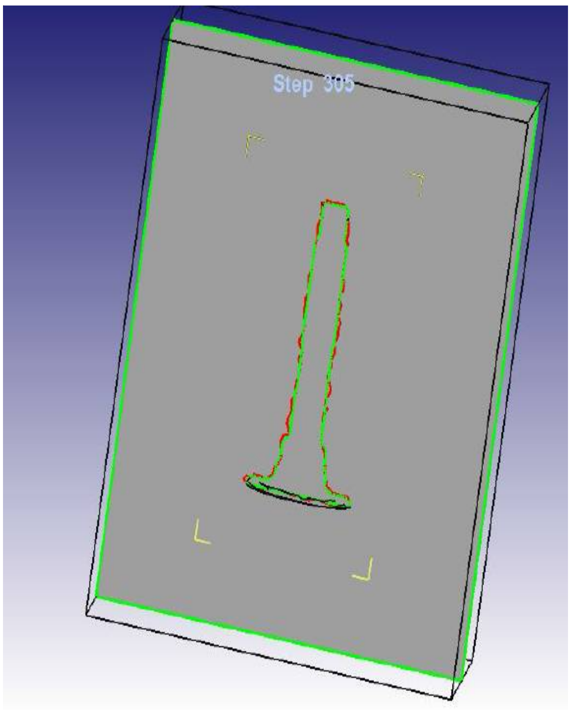
Load vs. Stroke Curve



Volume vs. Time



Slicing



Process Constraints

Flash thickness: 2

Flash width: 5.5

Co-efficient of friction: 0.3

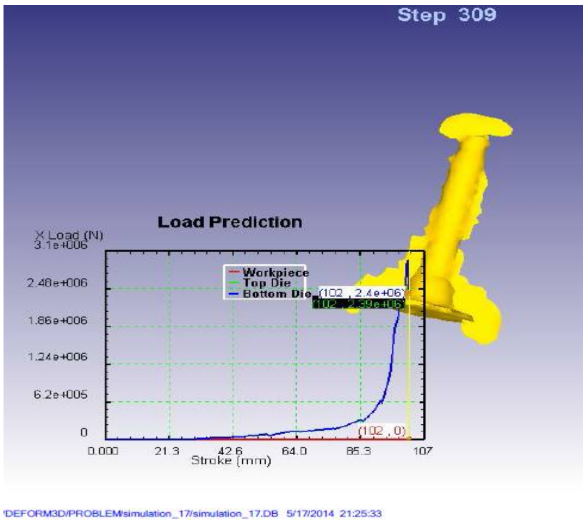
Die temperature: 400

Billet temperature: 1050

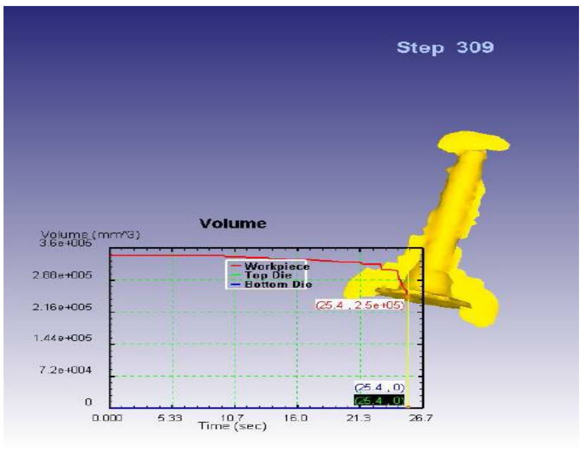
Complete filling: **Yes**

Simulation 17

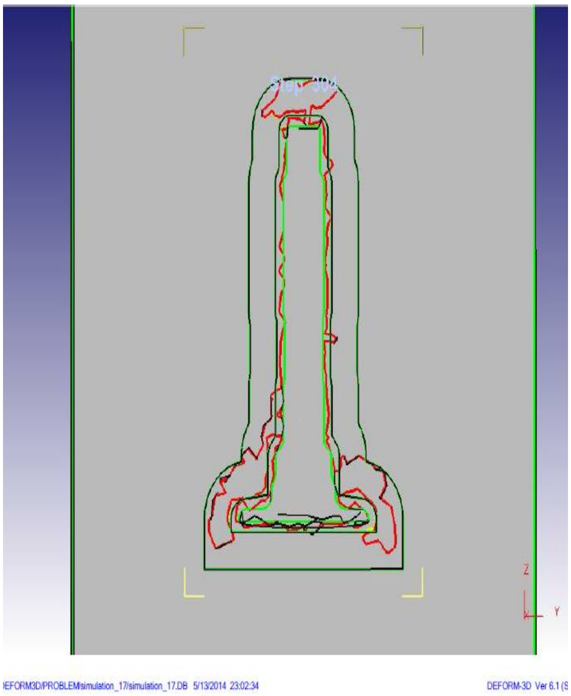
Load vs. Stroke Curve



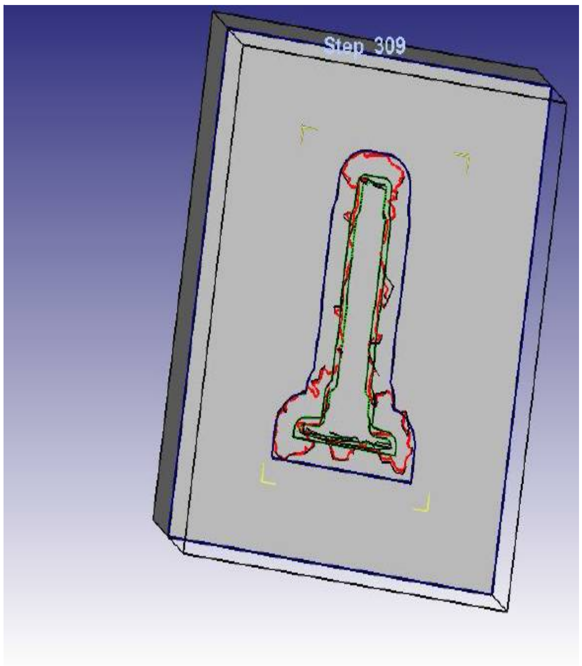
Volume vs. Time



Slicing

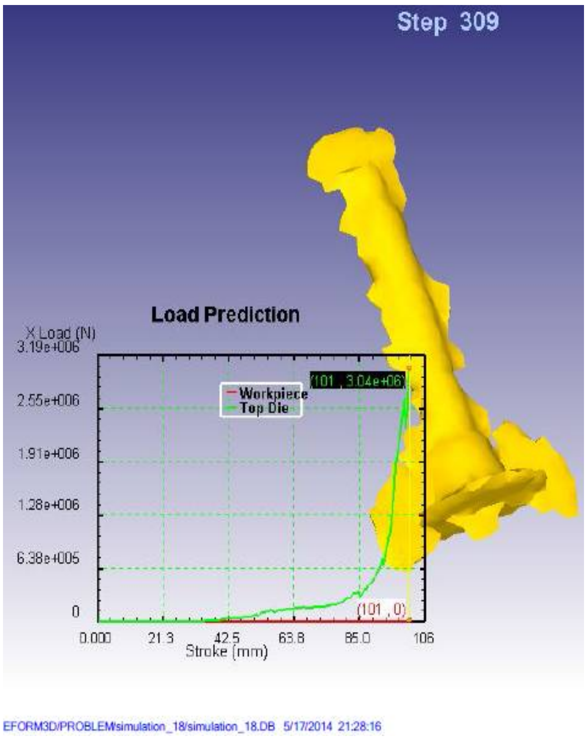


Slicing

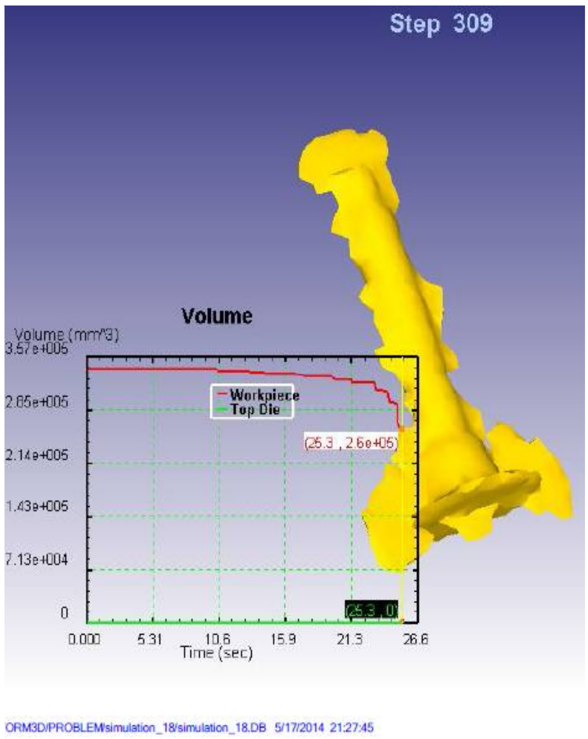


Simulation 18

Load vs. Stroke



Volume vs. Time



Process Constraints

Flash thickness: 2

Flash width: 6

Co-efficient of friction: 0.4

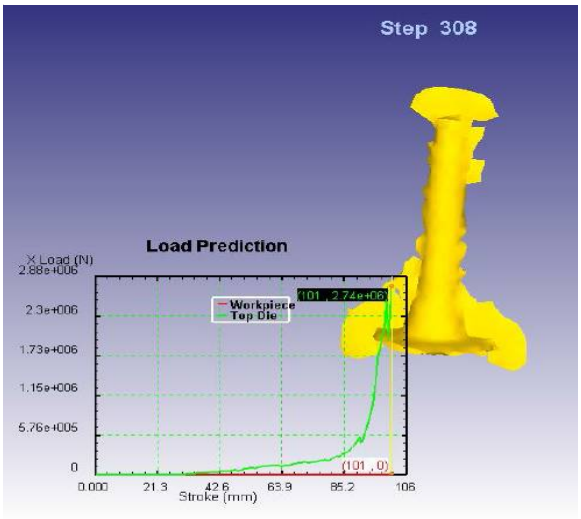
Die temperature: 200

Billet temperature: 1100

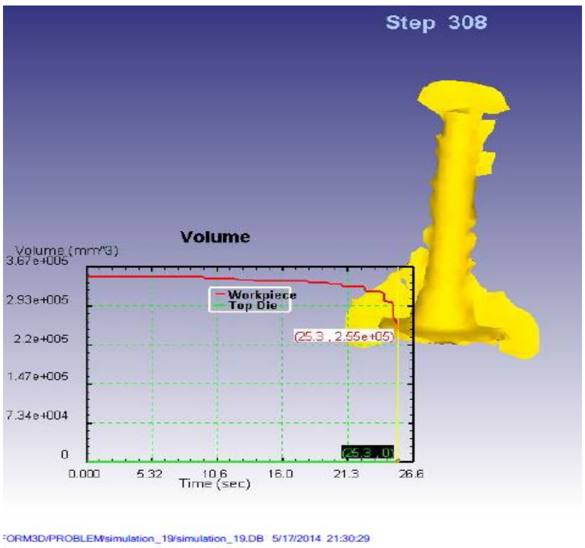
Complete filling: **Yes**

Simulation 19

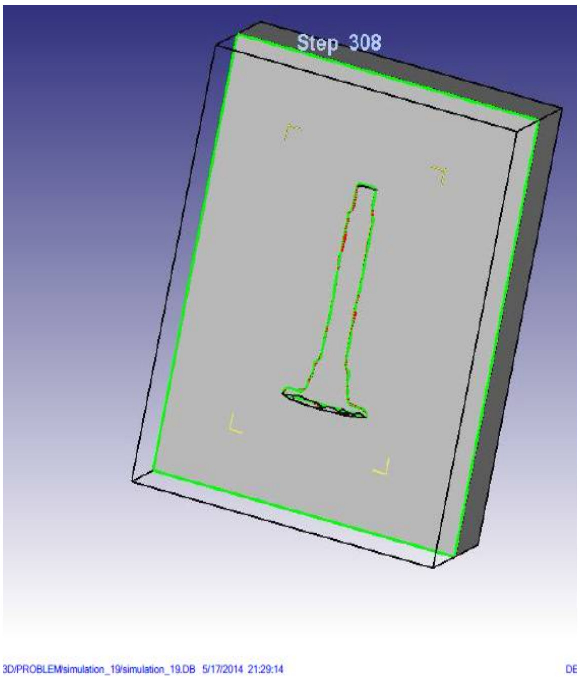
Load vs. Stroke



Volume vs. Time

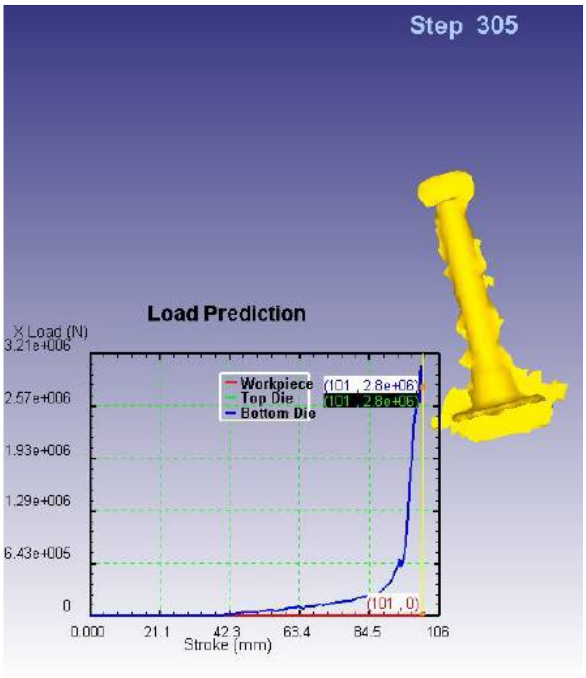


Slicing

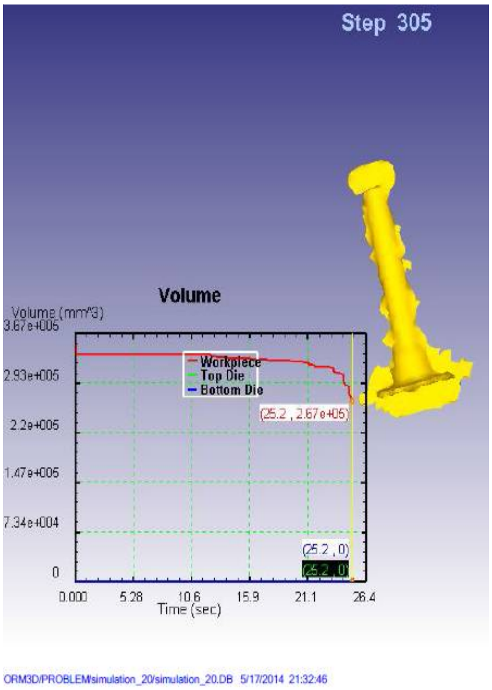


Simulation 20

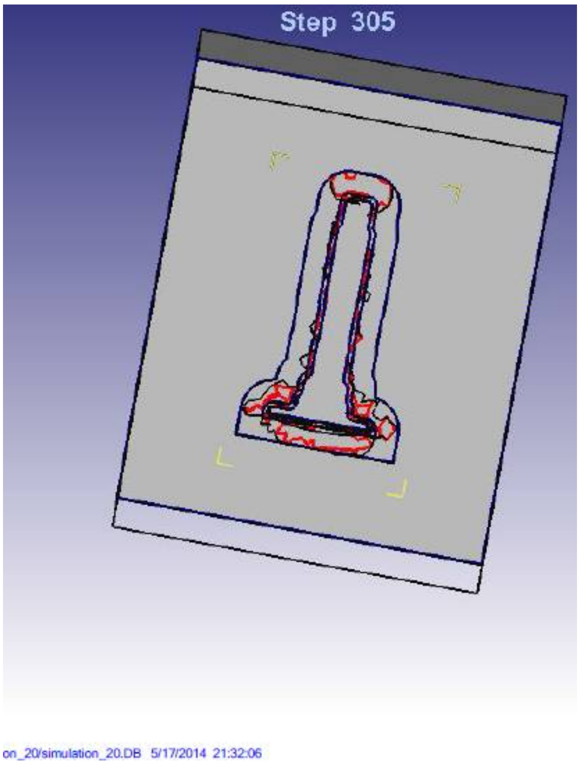
Load vs. Stroke



Volume vs. Time

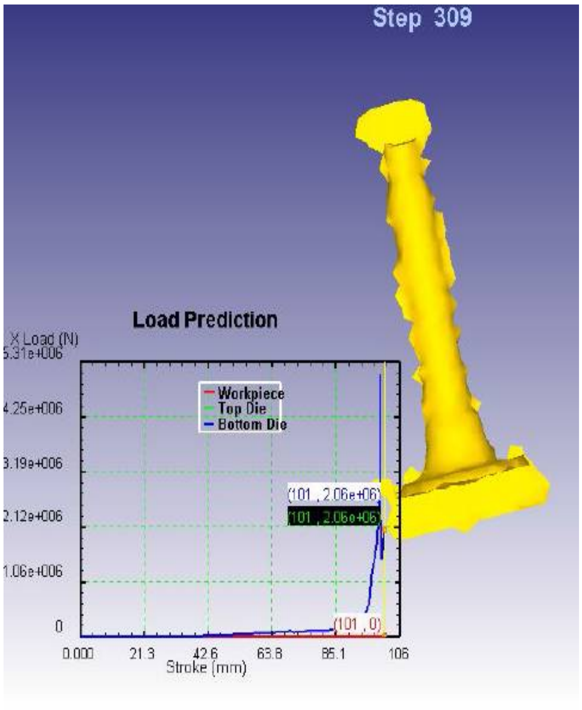


Load vs. Stroke Curve

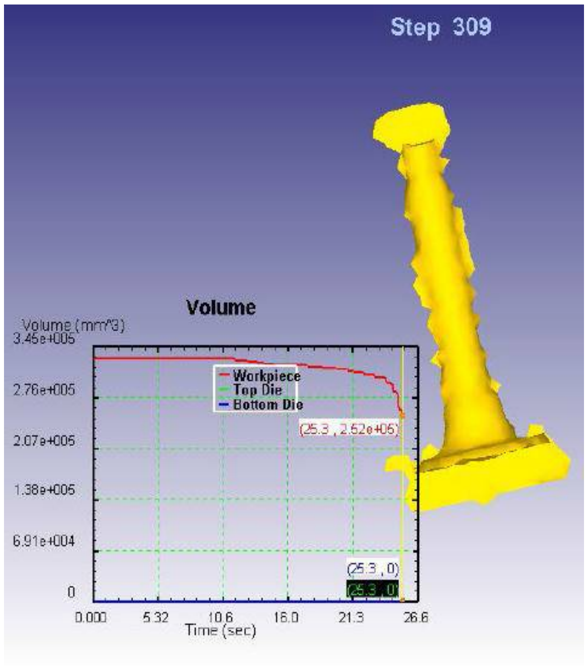


Simulation 22

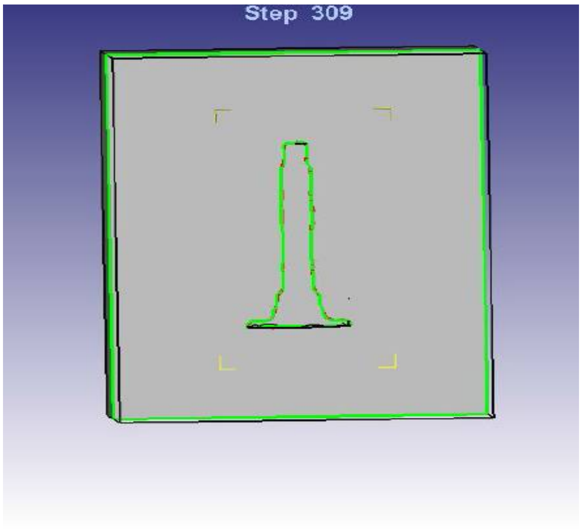
Load vs. Stroke



Volume vs. Time



Slicing



Process Constraints

Flash thickness: 2.5

Flash width: 5.5

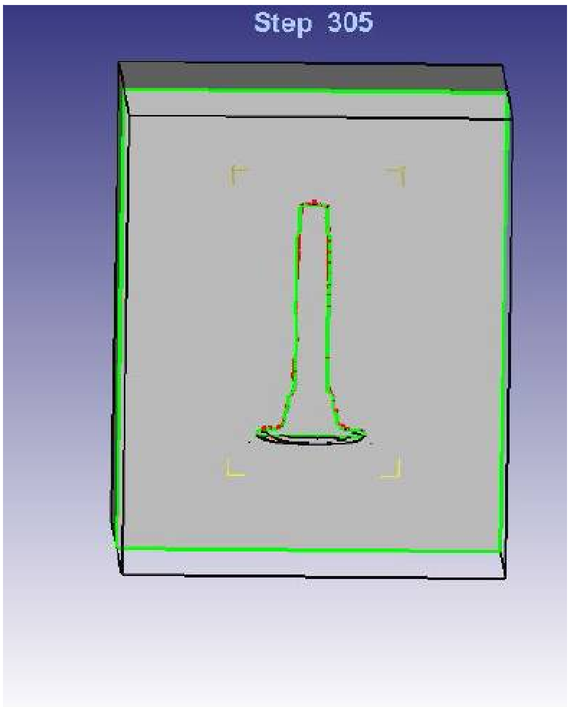
Co-efficient of friction: 0.2

Die temperature: 200

Billet temperature: 1150

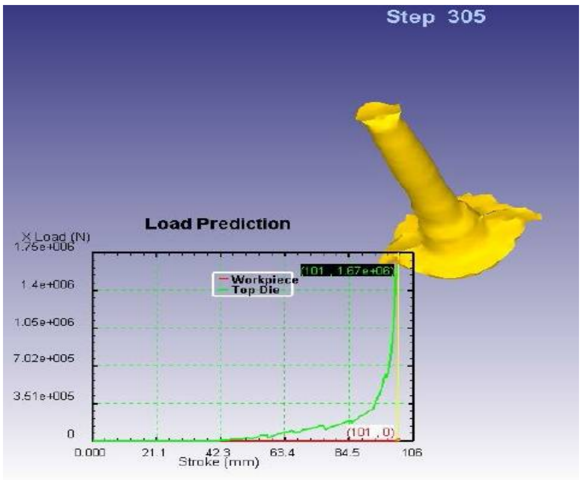
Complete filling: **Yes**

Load vs. Stroke Curve

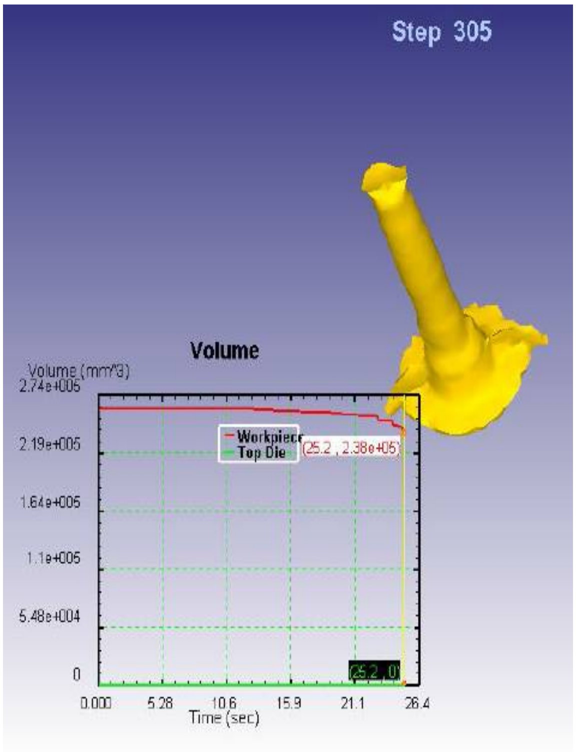


Simulation 24

Load vs. Stroke

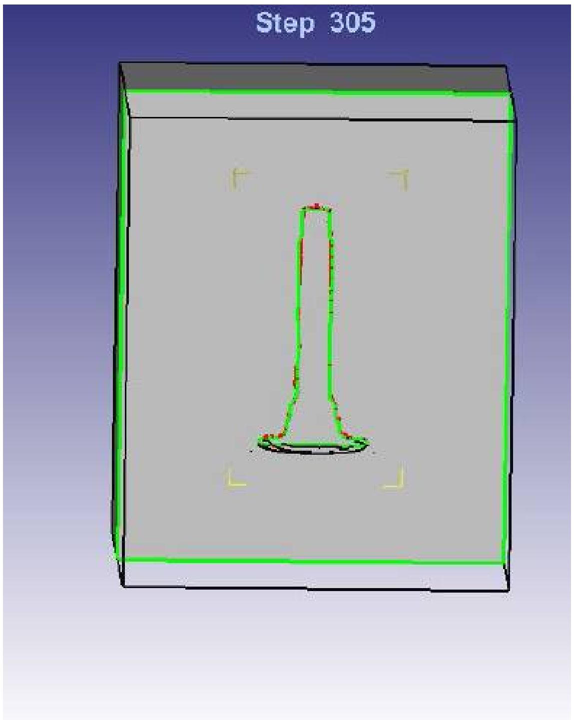


Volume vs. Time

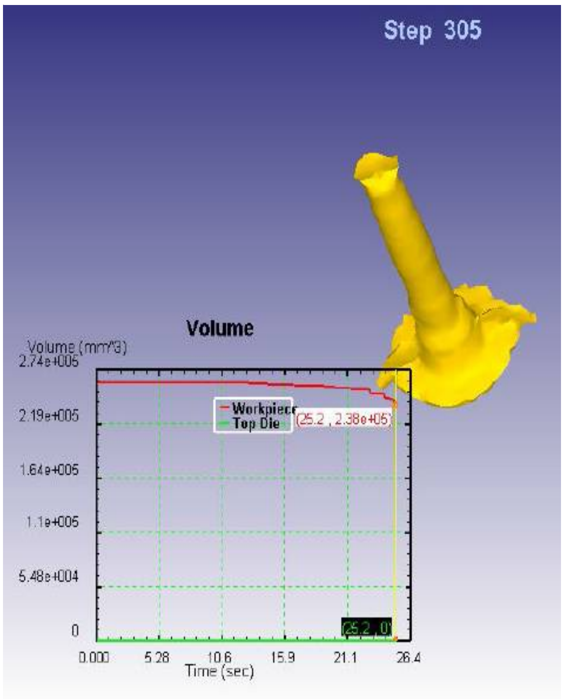


Simulation 25

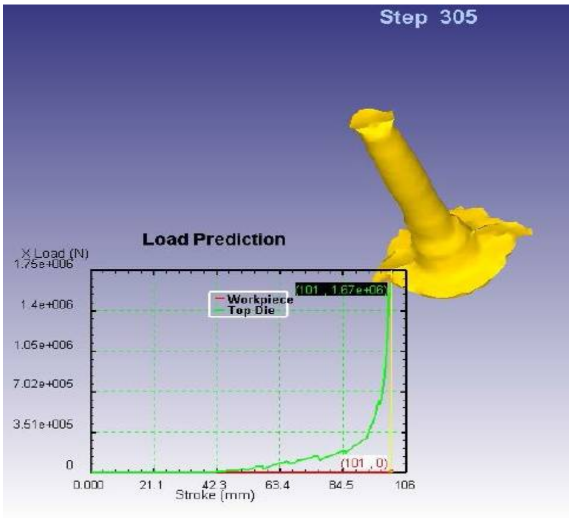
Slicing



Volume vs. Time



Load vs. Stroke Curve



Process Constraints

Flash thickness: 2.5

Flash width: 6

Co-efficient of friction: 0.2

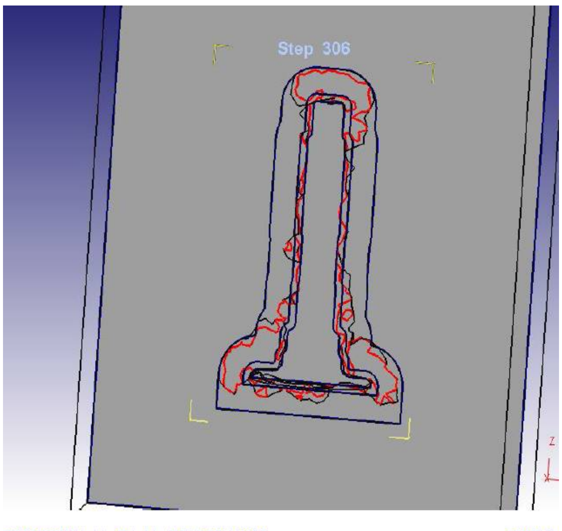
Die temperature: 300

Billet temperature: 1050

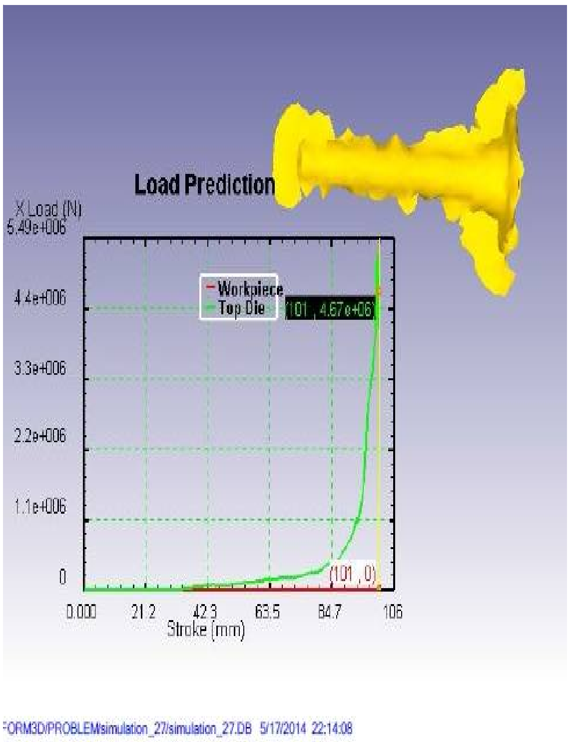
Complete filling: **Yes**

Simulation 27

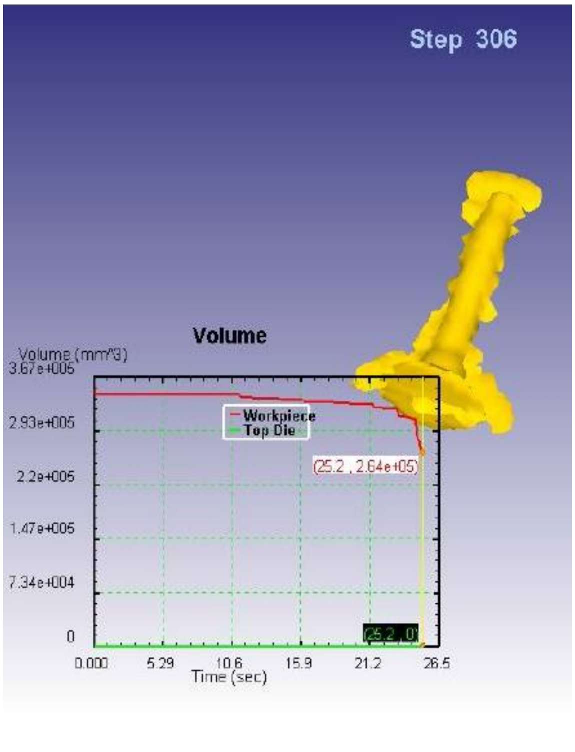
Slicing



Load vs. Stroke Curve

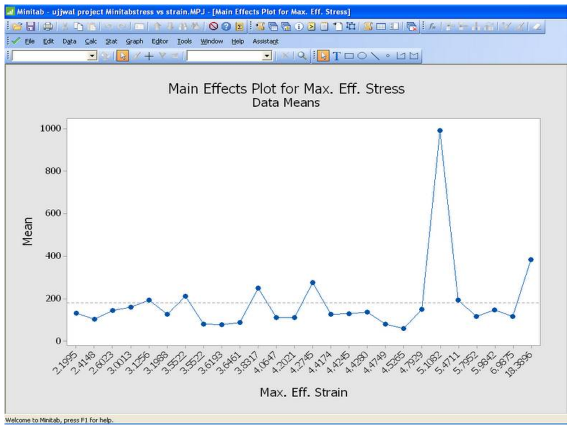


Volume vs. Time

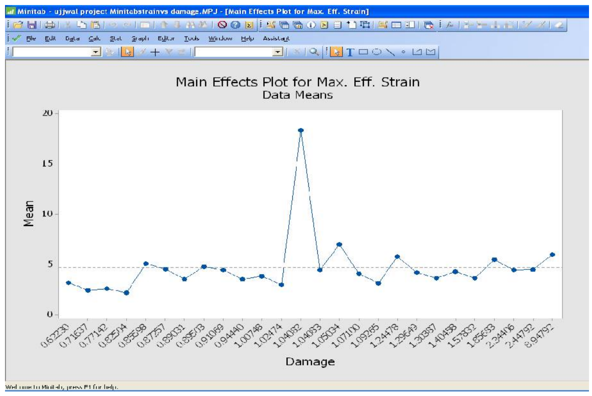


Damage

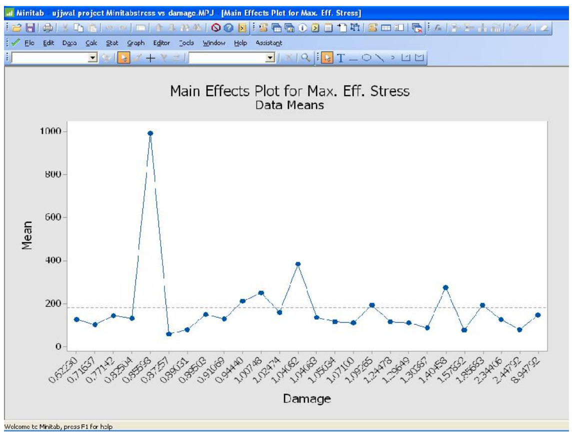
Graphical Representation of Maximum Effective Stress Vs. Maximum Effective Strain



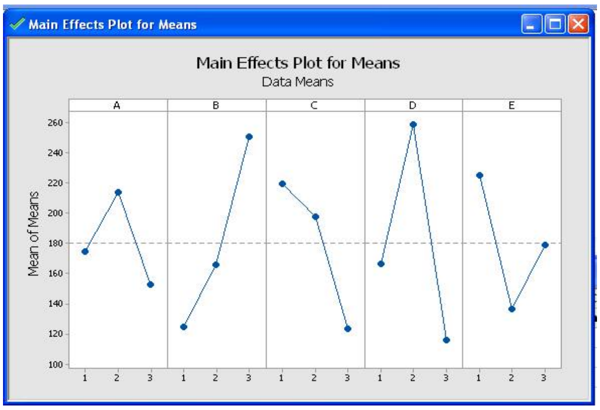
Graphical Representation of Maximum Effective Strain Vs. Damage



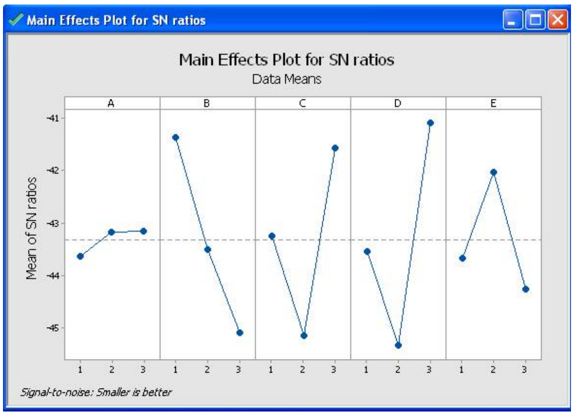
Graphical Representation of Maximum Effective Stress Vs. Damage



Graphical Representation of Main Effect Plot for Mean of Maximum Effective Stress



Graphical Representation of Main Effect Plot for SN Ratio of Maximum Effective Stress



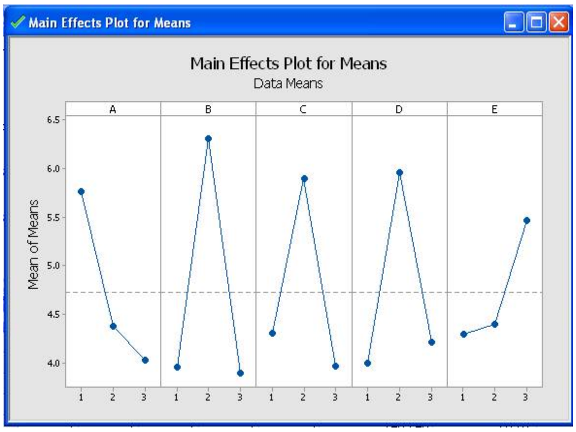
For SN ratio maximum is considered better result while for mean plot minimum is considered better.

From the above graph for minimum "maximum effective stress"

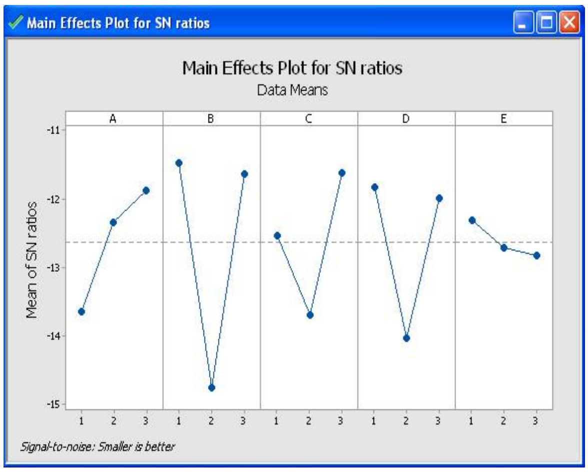
Constrains considered will be:

	Considered Parameters				
	A (C1)	B (C2)	C (C3)	D (C4)	E (C5)
For Mean plot	3	1	3	3	1
For SN ratio plot	3	1	3	3	1

Graphical Representation of Main Effect Plot for Mean of Maximum Effective Strain



Graphical Representation of Main Effect Plot for SN Ratio of Maximum Effective Strain

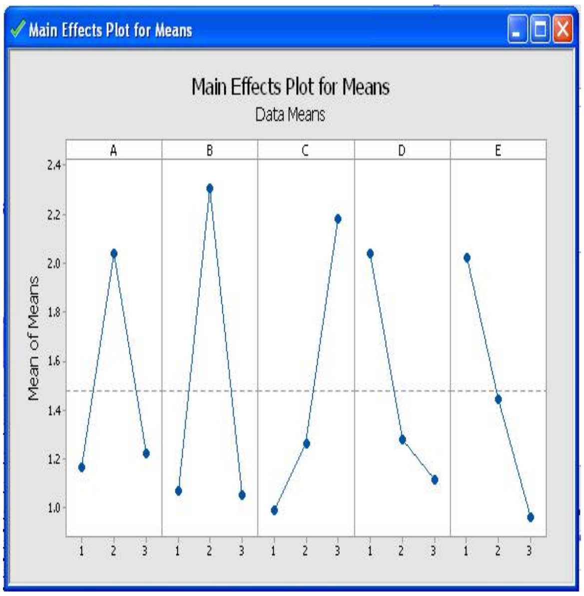


From the above graph for minimum "maximum effective strain"

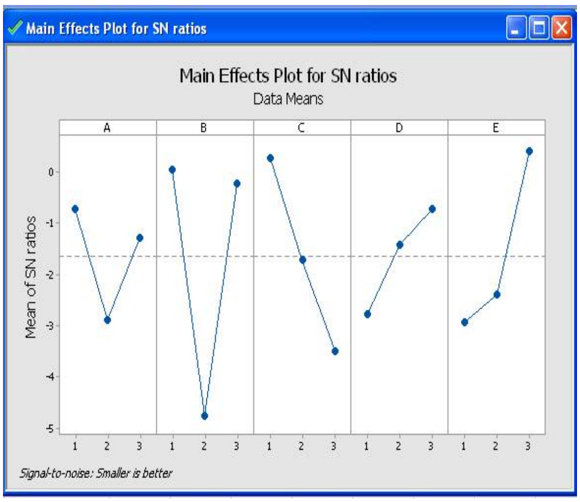
Constrains considered will be:

	Considered Parameters				
	A (C1)	B (C2)	C (C3)	D (C4)	E (C5)
For Mean plot	3	1	3	1	1
For SN ratio plot	3	1	3	1	1

Graphical Representation of Main Effect Plot for Mean of Damage



Graphical Representation of Main Effect Plot for SN Ratio of Damage



From the above graph for minimum "maximum effective strain"

Constrains considered will be:

	Considered Parameters				
	A (C1)	B (C2)	C (C3)	D (C4)	E (C5)
For Mean plot	1	1	1	3	3
For SN ratio plot	1	1	1	3	3

Results	Billet Temperature (^{0}C)	Die Temperature (^{0}C)	Flash Thickness (mm)	Flash Width (mm)
THEORETICAL	1100-1250 (*ASM Handbook* Vol. 14, p. 157)	205 -315 (*ASM Handbook* Vol. 14, p. 164)	1.525 (BY BRUCHANOV & REBELSKII formula)	6
SIMULATED	1150	300	1.5	6

CONCLUSION

Using deform for simulation makes the forging method relatively easier as the cost involved in die making, testing and rejecting due to minor or major complication or errors is eliminated to noticeable extent. This form of testing through simulation provides a wider verity of materials to be examined for being used as die material for dies. Faster rate of production better accuracy and even saving of materials used for production of products have been achieved by the implementation of this technology.

The project has been completed and considered constrains for minimizing "maximum effective stress, Maximum effective strain and Damage" has been shown in tabulated format.

We can conclude that flash width should be 4 times the flash thickness as all other literatures also reveals. Minimum value of Maximum Effective stress 75.8 & Maximum effective strain 1.57 & Damage being 3.619 for complete filling.

REFERENCES

1. 2nd International Conference on Engineering Optimization September 6 - 9, 2010, Lisbon, Portugal1 Pareto-based multi-objective hot forging optimization using a genetic algorithm Catarina F. Castro1, Carlos C. António2, Luisa C. Sousa3
2. Die design and die making practice, industrial press new York, 1989
3. Forging die design and practice by R saran S N Prasad and N P Saksena, s chand company, new delhi, published 1982
4. Forging industry handbook, forging industry association, capleveland, Ohio 1970
5. Materials Processing and Design: Modeling, Simulation and Applications : NUMIFORM 2004 : Proceedings of the 8th International Conference on Numerical Methods in Industrial Forming Processes : Columbus, Ohio, 13-17 June, 2004, Part 1.
6. Modelling of metal forming processes: proceedings of the Euromech 233 Colloquium, Sophia Antipolis, France, August
7. New concepts in die design physical and computer modeling applications Victor Vazquez*, Taylan Altan ERC for Net Shape Manufacturing, Ohio State University, Columbus, OH 43210, USA
8. Optimization of cold forging perform tools using Pseudo Inverse Approach Ali HALOUANI, Yu-ming LI, ABBES Boussad, Ying-qiao GUO, Received 28 August 2012; accepted 25 October 2012
9. Simulation Engineering Mikihiko Ohnari, IOS Press, 1998
10. Simulation of Material Processing: Theory, Methods and Application: Proceedings of the 7th International Conference NUMIFORM 2001, Toyohashi, Japan 18-21 June 2001
11. Simulation of Material Processing: Theory, Methods and Application: Proceedings of the 7th International Conference NUMIFORM 2001, Toyohashi, Japan 18-21 June 2001.