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 constrain.

**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
HORIZONTAL (X-AXIS) DIMENSIONS (WITH ALLOWANCES)

Total length: 234.6 mm

Calculation of plan area

\[ A \text{ (Plan area)} \]

Calculation for volume of the product

\[ V \text{ (From plan area)} \]

Calculations:

Vertical (y-axis) dimensions (with allowances)

\[ L_{eq} = 234.6 \]

Selection of Material

AISI 1045 STEEL (\( \rho = 7.86 \text{ gm/mm}^3 \))

WEIGHT = \( \rho v = 2.2 \text{ kg} \)

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.

\[ D_{\text{equivalent}} = 50 \text{ mm} \]

\[ L_{\text{equivalent}} = \sum L_i = \text{required length} \]

Calculation of Equivalent Length

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

\[ \frac{\pi}{4}d^2 L_1 = \frac{\pi}{4}(50)^2 L_2 \]

Length of the Stock

\[ \sum L_i = 139.87 \text{ mm} \]

Selection of Draft Angle

For depth/height (\( H \))

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

<table>
<thead>
<tr>
<th>Dimensions</th>
<th>Formulae</th>
<th>Calculation</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flash thickness (t)</td>
<td>( T = 0.015 \sqrt{A} )</td>
<td>0.015 ( \sqrt{8492} )</td>
<td>1.38 mm</td>
</tr>
<tr>
<td>Flash width(b)</td>
<td>( b = 4 \times t )</td>
<td>( b = 4 \times 1.38 )</td>
<td>5.52 mm</td>
</tr>
<tr>
<td>Gutter thickness(g)</td>
<td>( g = 3 \times t )</td>
<td>( b = 3 \times 1.38 )</td>
<td>4.14 mm</td>
</tr>
<tr>
<td>Gutter width(b1)</td>
<td>( b1 = 4 \times b )</td>
<td>4 ( \times 1.14 )</td>
<td>22.08 mm</td>
</tr>
</tbody>
</table>
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 \( \frac{1}{4} \) 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 \( \text{into} \) 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 \( \text{[ ]} \). 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 **Process Monitor**

**Simulation Graphics**

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

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

### Variable Considered Simulation Wise (Taguchi Experimental Design)

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

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

Simulation 6

Load vs. Stroke Curve

Volume vs. Time
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**

**Simulation 14**

**Load vs. Stroke Curve**

**Volume vs. Time**
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
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

Simulation 20

Load vs. Stroke
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

Simulation 24
### 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

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
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:

<table>
<thead>
<tr>
<th>Considered Parameters</th>
<th>A (C1)</th>
<th>B (C2)</th>
<th>C (C3)</th>
<th>D (C4)</th>
<th>E (C5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>For Mean plot</td>
<td>3</td>
<td>1</td>
<td>3</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td>For SN ratio plot</td>
<td>3</td>
<td>1</td>
<td>3</td>
<td>3</td>
<td>1</td>
</tr>
</tbody>
</table>

From the above graph for minimum “maximum effective strain”

Constrains considered will be:

<table>
<thead>
<tr>
<th>Considered Parameters</th>
<th>A (C1)</th>
<th>B (C2)</th>
<th>C (C3)</th>
<th>D (C4)</th>
<th>E (C5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>For Mean plot</td>
<td>3</td>
<td>1</td>
<td>3</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>For SN ratio plot</td>
<td>3</td>
<td>1</td>
<td>3</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
From the above graph for minimum “maximum effective strain”
Constrains considered will be:

<table>
<thead>
<tr>
<th>Considered Parameters</th>
<th>A (C1)</th>
<th>B (C2)</th>
<th>C (C3)</th>
<th>D (C4)</th>
<th>E (C5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>For Mean plot</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>For SN ratio plot</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>3</td>
<td>3</td>
</tr>
</tbody>
</table>

**RESULTS**

<table>
<thead>
<tr>
<th>Results</th>
<th>Billet Temperature (°C)</th>
<th>Die Temperature (°C)</th>
<th>Flash Thickness (mm)</th>
<th>Flash Width (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIMULATED</td>
<td>1150</td>
<td>300</td>
<td>1.5</td>
<td>6</td>
</tr>
</tbody>
</table>

**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


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


