Analysis of the thermo-mechanical deformations in a hot forging tool by numerical simulation

R.L-Cancelos¹, F. Varas², E. Martín³, I. Viéitez²

¹Departamento de Ingeniería de los Materiales, Mecánica Aplicada y construcción. E. de Ing. Industrial, Universidad de Vigo, Campus Marcosende, E-36310 Vigo
²Departamento de Matemática Aplicada II. E. de Ing. de Telecomunicación, Universidad de Vigo, Campus Marcosende, E-36310 Vigo
³Departamento de Ingeniería Mecánica, Máquinas y Motores Térmicos y Fluidos. E. de Ing. Industrial, Universidad de Vigo, Campus Marcosende, E-36310 Vigo

E-mail: rlopezcanelos@uvigo.es

Abstract. Although programs have been developed for the design of tools for hot forging, its design is still largely based on the experience of the tool maker. This obliges to build some test matrices and correct their errors to minimize distortions in the forged piece. This phase prior to mass production consumes time and material resources, which makes the final product more expensive.

The forging tools are usually constituted by various parts made of different grades of steel, which in turn have different mechanical properties and therefore suffer different degrees of strain. Furthermore, the tools used in the hot forging are exposed to a thermal field that also induces strain or stress based on the degree of confinement of the piece. Therefore, the mechanical behaviour of the assembly is determined by the contact between the different pieces.

The numerical simulation allows to analyse different configurations and anticipate possible defects before tool making, thus, reducing the costs of this preliminary phase.

In order to improve the dimensional quality of the manufactured parts, the work presented here focuses on the application of a numerical model to a hot forging manufacturing process in order to predict the areas of the forging die subjected to large deformations.

The thermo-mechanical model developed and implemented with free software (Code-Aster) includes the strains of thermal origin, strains during forge impact and contact effects. The numerical results are validated with experimental measurements in a tooling set that produces forged crankshafts for the automotive industry. The numerical results show good agreement with the experimental tests. Thereby, a very useful tool for the design of tooling sets for hot forging is achieved.

1. Introduction
Nowadays forging process remains a widely used technology [1]. Since the development of computers and their diffusion in companies of metal forming, there have been many researchers and developers who have designed models and software capable of simulating processes of forging [2].

Within the broad spectrum covered by forging problems, some researchers have focused on evaluating the elastic deformation occurring in forging tools by the method of finite element. One of
the early work in this field is done by Lange et al. [3], where an analysis of the causes of failure of forging dies due to fatigue phenomena, with certain simplifications assumed (as an axisymmetric problem is carried out). A further work is published by Nagao et al. [4], they perform an analysis of the states of stress that occur in the inserts during the forging operation for automotive industry, also solved for an axisymmetric problem. Qin and Balendra [5] study the influence of the elasticity of the die relative to the final dimensions of the part in the case of extrusion forging.

The work presented by Joun et al. [6] focuses on the analysis of sequence control for hot forging considering the behavior of the die as infinitely rigid. With this simplification Joun et al. [7] studied the elastic behavior of the die in cold forging applications. First, they calculated the forging pressure under the assumption of die infinitely rigid. Then, the calculated pressures are imposed on the die modeling its elastic behavior. They also considered the various parts of the tool and applied a penalization algorithm to solve the contact.

Ou and Armstrong [8] study the important influence of the elasticity of the forging die and the thermal distortion in the process of manufacturing a titanium turbine blade for the aeronautic industry. But they only focus on the die, they do not include all the set tool and the geometry of the piece is relatively simple.

Jun et al. [9] present a work with a practical approach to systematically estimate the geometric dimensions of pieces manufactured by cold forging. They make the assumption that the matrices are rigid to calculate the forging stress. The characteristics of the studied process allow them to use an axisymmetric model.

In the present work a valid model for the evaluation of induced stress and strain in hot forging dies is solved, considering the elastic behavior of the tool and the influence of the thermal field. The model is validated by acquiring data during the manufacture of crankshafts for the automotive industry

2. Description of the process
The shaping of crankshafts is performed in a 4000T mechanical press in two stages. Therefore two pairs of matrices are used (called prior and final die). The control operation is semiautomatic and it is performed by an operator. Between each forged crankshaft two operations are performed, the cooling and the lubrication of the surfaces of the dies.

The forging tool is formed of two parts, the upper and the lower, being both symmetric. This study focuses on the behavior of the lower part because it is the one exposed to a larger thermal field.

In figure 1 a simplified scheme of the lower part used by CIE Galfor S.A. company is shown. Geometrical details are not shown for confidentiality reasons. This tool is formed by 12 pieces built in different steel qualities. The set broadly contains a holder (P1), a support slab (P2), the dies of the first and second press stroke (D1 and D2 respectively) and a third lateral block (P3) that fills in the empty space. Most of the pieces simply rest lying ones on the others and are fixed by means of elastic joints named disc springs that are responsible for the compression of the set D1, D3 and P3. These joints are located between the wedges (P6-1, P6-2, P6-3 and P7) and the dies and the piece P3.

Following European codification (EN 10027/2:1992) the dies D1 and D2 are made of steel W Nr 1.2344 (table 1 shows the physical properties and their dependency with the temperature), the piece P1 of W Nr 1.2738 (table 1) and the rest of W Nr 1.2714 (table 1).

| Steel  | T (°C) | E (GPa) | \( \rho \) (\( \text{kg/m}^3 \)) | \( \lambda \) (\( \text{W/m} \cdot \text{°C} \)) | c (\( \text{J/kg \cdot K} \)) | \( \alpha \) (\( \text{mm/mm \cdot °C} \)) |
|--------|--------|---------|------------------|-----------------|-----------------|-----------------|
| 1.2344 | 20     | 210     | 7800             | 25              | 460             | 11.0            |
|        | 300    | 188     | 7726             | 27.9            | 526             | 11.8            |
| 1.2738 | 20     | 210     | 7800             | 25              | 460             | 12.8            |
|        | 300    | 193     | 7730             | 33.4            | 557             | 13.8            |
| 1.2714 | 20     | 210     | 7800             | 25              | 460             | 12.5            |
| 300 | 194 | 7710 | 36.7 | 513 | 13.5 |

**Figure 1.** Schematic representation of the lower tool. D1 and D2 indicate the position of the dies of the first and second press stroke respectively.

A picture of the lower part tool in its work position is shown in figure 2.

**Figure 2.** Lower part tool located in its work position.

### 3. Implemented model

The nature of the physical problem requires solving two models: a thermal and a mechanical model. Therefore the model presented solves the coupling between the thermal problem and thermo-mechanical as a one-way problem. This simplification is done because it is considered that there is no heat generation on the contact surfaces and the opening of the contact due to the press strokes does not influence the heat transfer between the different pieces because the characteristic stroke time is very small (0.1s).

#### 3.1. Thermal Model

Although the problem is essentially a transitory one, the resolution of the thermal problem arises as a steady state (eq 1) because the forging characteristic time is much shorter than the thermal inertia.
\[ 0 = \text{div} \left( k \nabla T \right) \text{ in } \Omega \]  

The heat source for the tool set are the lower die surfaces (that remain some time in contact with the hot billet). To model its effect thermographies (figure 3 and 4) are used. Therefore the temperature distribution of the die surfaces are imposed as a Dirichlet boundary condition (eq 2).

\[ T = T_0 \text{ on } \Gamma_D \]  

![Figure 3. Thermography of prior die.](image)

![Figure 4. Thermography of final die.](image)

The internal surfaces that are not in contact with others are considered as insulated (eq 3).

\[ k \frac{\partial T}{\partial n} = 0 \text{ on } \Gamma_I \]  

As observed in figure 2, the front of the tool is exposed to the environment. Therefore the heat flux through the boundary is solved by a model of convection (eq 4) to the environment. The value of the coefficient \( h \) is 10 \( \frac{W}{m^2 \cdot k} \) (see Incropera [10] and Chao et al. [11]) and the temperature of the environment is 20°C.

\[ k \frac{\partial T}{\partial n} = h \left( T_\infty - T \right) \text{ on } \Gamma_f \]  

The rest of the external boundaries are in contact with the press. The strategy for modeling the heat transfer is performed by using an equivalent heat transfer coefficient (eq 5) Cooper et al. [12]. The value used is 40 \( \frac{W}{m^2 \cdot k} \) [13] while the selected temperature of the heat sink for the press foundation is \( T_\infty = 15°C \).

\[ k \frac{\partial T}{\partial n} = h_{eq} \left( T_\infty - T \right) \text{ on } \Gamma_f \]  

3.2. Mechanical Model

As the nature of the deformation is elastic it is assumed that the behavior of different materials is elastic and isotropic (as Jun et al [9]), being the material properties variable with temperature (see table 1).

As the tool is constituted with several pieces, the surfaces that come in to contact must fulfill equation 6. To model the contact between the different parts a penalization algorithm is used as in the work of Jou et al [7]. The total number of contacts are 35, 16 in x direction, 10 in y direction and 9 in z direction.

\[ \bar{\sigma}^i(\bar{u}) \bar{n}_i = p_N \bar{n}_i + p_T \bar{t}_i \text{ in } \Gamma^i \]  

where \( \Gamma^i \) are the surface boundaries of each piece that comes into contact with the surface of a different one. \( p_N \) stands for the normal stress in the contact surfaces while \( p_T \) represents the tangential stresses.

The final form of the forging piece is conditioned by the shaping of the footprint at the moment of maximum load. Thus, the mechanical problem is solved for the instant under the maximum load.
addition, it is also necessary to decouple the problem of the forged piece from the elastic deformation of the tool due to the size of the problem and the computational cost. This procedure has been used in previous works like Jun et al. [9].

The calculation of forging stress (see eq. 7) under the assumption of rigid die was made by the company through the use of the commercial software FORGE3© (see results on figure 5). The imposed stress on the final die is described by the equation 7 as a boundary condition of Neumann type.

\[ \vec{\sigma}(\vec{u}) \vec{n} = \vec{f}^{FOR} \text{ in } \Gamma^D \]

where \( \Gamma^D \) is the footprint surface of the die and \( f^{FOR} \) is the known forge surface stress.

**Figure 5.** Stresses field over the longitudinal section of die D2 obtained with FORGE3®.

The lower boundary of the tool is placed, with no restriction, over a steel surface. Thus, this surface can expand in x and y directions (in order to manipulate the tool there is a gap in y direction). To mimic this behavior the movement is only blocked in z direction (eq. 8). In order to prevent an ill-posed problem it is necessary to include certain reactions in x and y directions in the base of the tool that permit the thermal expansion but that do not alter the mechanical problem. This is achieved by including in the model very low stiffness springs in the nodes of the tool base. These springs are modeled as nodal discrete elements without dimensions (0D).

\[ u^{P1}_z = 0 \text{ in } \Gamma^{P1}_u \]

Because of the type of fixing between the back part of the tool and the outer piece, this part is modeled assuming no movement in the x direction, and then, simulating the contact problem between the tool and the exterior.

\[ \vec{u}^{P1} = 0 \text{ in } \Gamma^{P1(-x)}_u \]

In all the wedges used in the tool, certain disc springs are placed for the elastic coupling between the wedges and the dies D1 and D2 and the supplementary piece P3, being in charge of absorbing the different expansions between the pieces while keeping them at work position. Due to the difficulty that a detailed numerical model for the disc springs entails, a simplification was done, modeling each disc spring as a cylindrical ring with an equivalent elastic module (dependent on the temperature) that reproduce the same rigidity than the real disc springs, whose values are indicated in table 2. The equivalent elastic module is obtained using the real force exerted on the set and its displacement.

**Table 2.** Equivalent elastic module (as a function of temperature) of the disc springs located between the wedges and the dies.

| T (°C) | \( E_{equil} \left( \frac{kN}{mm^2} \right) \) |
|--------|-----------------------------------------------|
| 20     | 82                                            |
| 300    | 75                                            |
Each wedge is fixed to the group through two bolts. Due to the reduced dimensions of the bolts in relation to the complete tool, the elements are not taken into account in the geometrical model but the forces induced by them are. The pressing force in each bolt is of 287N (estimated taking into account the contact surface of the head of the bolts).

4. Experiment description
To evaluate the mechanical behavior of the final die is decided to monitor its base due to the impossibility of placing sensors on the surface of the footprint. The physical quantities evaluated are the unitary strains and the temperature. For this purpose two types of sensors are used, 8 self-compensated strain gauges (G1 to G8) and 7 thermocouples (T1 to T7) type k. To accommodate the sensors it is necessary to perform a machining of the base die. In figure 6 it is shown the position of the sensors on the basis of the final die (D2).

![Figure 6. Sensors position on the base of the final die (D2).](image)

The temperature values are used to validate the thermal problem and also to correct the strain signals registered as explained in Watson [14] due to the thermal output.

The even gauges recorded the longitudinal strain while the odd gauges the transversal one. The gauges G1 to G5 are positioned at the central zone of the base and the G6 to G8 at an extreme. The signal of the gauge G5 was lost at the beginning of the experiment.

Data acquisition was performed during the manufacturing of a series of crankshafts, and a total of forty forging strokes for the final die were recorded.

5. Experimental and numerical results for the thermal problem
Figure 7 shows the temperature value for each thermocouple and each final forging stroke. It can be observed the temperature variation and the absence of a defined pattern.

The strong variability of the process makes necessary to determine the temperature field by a representative mean value, so a statistical treatment of the data is performed. The results of this statistical analysis are shown in Figure 8 using a box plot for each thermocouple. The maximum temperatures were recorded for T4 and T5 thermocouples (188°C). The measures for the thermocouple T5 had the highest variability. The thermocouples of the center zone (T1-T3) recorded a temperature around 165°C, while the thermocouples of the extreme zone recorded the minimum average temperature, around 157°C.
Figure 7. Signal recorded by thermocouples for each final forging stroke.

The numerical result of the temperature field inside the forging tool is shown in figure 9. The highest temperature values are located in the dies area and decrease toward the external boundaries.

Figure 8 shows also the numerical values extracted from the probe points located on the same coordinates as the thermocouples, being the values represented by blue blades. Although there are some differences between numerical results and the mean temperature, all numerical values are within the range of temperatures recorded for each thermocouple. Therefore it is considered that the proposed thermal model reproduces the recorded average field and is used to evaluate the thermal contribution to the mechanical problem.

Figure 8. Comparison between temperature measurements registered (box plot) and numerical values (blue blades).

Figure 9. Thermal field simulation result for the thermal field.

6. Experimental and numerical results for the mechanical problem

The data recorded and corrected by the strain gauges are shown in figure 10 for the forty press strokes. The corrections are made based on the thermal output and the free expansion [15]. The last one is estimated based on the minimum temperature value extracted from the result of the thermal problem (62.70°C) and the coefficient of thermal expansion of the steel. The free expansion estimated value is 470µε.

For each signal recorded from each strain gauge, two types of strain can be extract. The basis signal corresponds to the strain of thermal origin and the peak signal corresponds to the maximum strain at the time of maximum load. The recorded data is processed statistically just as in the case of temperature data. Then, two average fields were calculated, the strain corresponding to the thermal field and the strain corresponding to the thermal field and the press stroke. These results are represented by a box plot, in figure 11, that includes the results for the thermal strain, and in figure 12 that shows the results for the thermal strain and the strain produced by the press stroke.
Mean values of the thermal deformation for each gauge have a similar value, around 1650µε and it is similar in the central zone and in the extreme. As for the values corresponding to blow forging are also similar between zones, except for the recorded values of the gauge G7. These decrease their values with respect to strain of thermal origin. On the other hand, the recorded values of the even gauges are greater than the values of the odd gauges. Therefore, transverse strain of the die is greater than the longitudinal. The value of the thermal deformation represents a third of total deformation.

Figure 10. Register of the corrected measures of the strain gauges for the experimental data.

![Figure 10](image10)

Figure 11. Strain of thermal origin for the experimental data and simulated results.

Figure 12. Strain at the instant of maximum load for the experimental data and simulation results.

Figures 11 and 12 also includes the numerical result of the simulation (represented by blades). The results for the strain of thermal origin in the probe points are similar to the values recorded. The values of G1, G3, G4, G6 and G8 are within the range of measured values. The largest relative errors correspond to the calculated values for G2 and G7 (14% y 17% respectively).
As for the results at the instant of maximum load, the calculated values for the gauges G1 and G3 are those falling within the measured range. The biggest relative errors correspond to the results for the G6 and G8 probe points (26% y 24% respectively), followed by G2 and G4 (20% y 18% respectively). The numerical values corresponding to the central zone are similar. This situation is inherited from forging stress calculations on the assumption of rigid die resulting in a homogeneous efforts on the central die.

Despite the difference between the measured and calculated values for both cases (thermal and thermo-mechanical) it is considered that the numerical result is optimal, because the relative errors calculated are less than 26%. This validates the proposed model.

Focusing now on the numerical results of the validated model, figure 13 shows the calculated displacement for the set tool along two transverse cuts. The positive displacement of the prior die is due to the thermal expansion. The final die is placed at the top center of the assembly and it is shown individually in figure 14.

![Figure 13. Displacement simulation result of the set tool (x50).](image1)

![Figure 14. Displacement simulation result of the final die (x50).](image2)

Figure 15 compares thermal and mechanical deformations represented by a longitudinal section of the surface of the final footprint. Thus, based on the results, the contribution of the thermal field to the total strain field is significant and it must be taken into account when evaluating the mechanical behavior of the tools used in hot forging.

![Figure 15. Comparison between displacement of thermal origin and thermo-mechanical (x50).](image3)

7. Conclusions
This paper presents a model to estimate the strain that occur in hot forging tools and it is validated with experimental measures obtained in a hot forging manufacturing process.

From the experimental results, it can be concluded that the thermal field does not show a fully defined pattern, there are fluctuations throughout the production process. This indicates that the thermal problem is very complex but making a series of careful simplifications it can be calculated an averaged thermal field representative of the real problem. The proposed thermal model reproduces the
recorded average field and, in consequence, is used to evaluate the thermal contribution to the mechanical problem.

There are also fluctuations in the strain field but these fluctuations cannot be directly related to the fluctuations of the thermal field. There are many dependent factors besides temperature such as the temperature of the steel billet, the degree of lubrication of the die, positioning parts, etc. That are not taken into account due to their complexity.

Currently the design process of forging dies is a costly process that consists of several test phases. The model proposed in this paper intends to reduce the design phase and therefore its costs.

8. Acknowledgement
This work has been carried out in collaboration with company CIE Galfor S.A. (Ourense, Spain) and was partially supported by funds of Xunta de Galicia (Galicia, Spain) and CDTI (Ministerio de Economía e Industria, Spain) under the project from the “Design and Industrial Productions” program, code 10DPI126E and MTM2013-47800-C2-2P.

References
[1] T. Takami, Production engineering strategies and metalworking at Toyota motor corporation, Procedia Engineering 81 (2014), 5-17.
[2] P. Hartley, I. Pillinger, Numerical simulation of the forging process, Computer Methods in Applied Mechanics and Engineering 195 (48) (2006), 6676-6690.
[3] K. Lange, A. Hettig, M. Knoerr, Increasing tool life in cold forging through advanced design and tool manufacturing techniques, Journal of Materials Processing Technology 35 (3) (1992), 495-513.
[4] Y. Nagao, M. Knoerr, T. Altan, Improvement of tool life in cold forging of complex automotive parts, Journal of Materials Processing Technology 46 (1) (1994), 73-85.
[5] Y. Qin, R. Balendra, FEM simulation of the influence of die-elasticity on component dimensions in forward extrusion. International Journal of Machine Tools and Manufacture 37 (2) (1997), 183-192.
[6] M. Joun, H. Moon, R. Shivpuri, Automatic simulation of a sequence of hot-former forging processes by a rigid-thermoviscoplastic finite element method. Journal of Engineering Materials and Technology 120 (4) (1998), 291-296.
[7] M. Joun, M. Lee, J. Park, Finite element analysis of prestressed die set in cold forging. International Journal of Machine Tools and Manufacture 42 (11) (2002), 1213-1222.
[8] H. Ou, C. Armstrong, Die shape compensation in hot forging of titanium aerofoil sections. Journal of Materials Processing Technology 125 (2002), 347-352.
[9] B. Jun, S. Kang, M. Lee, R. Park, M. Joun, Prediction of geometric dimensions for cold forgings using the finite element method. Journal of Materials Processing Technology 189 (1), (2007) 459-465.
[10] F. P. Incropera, Introduction to heat transfer. John Wiley & Sons. (2011)
[11] Y. J. Chao, X. Qi, W. Tang, Heat transfer in friction stir welding-experimental and numerical studies. Journal of Manufacturing Science and engineering 125 (1) (2003), 138-145.
[12] M. Cooper, B. Mikic, M. Yovanovich, Thermal contact conductance. International Journal of Heat and Mass Transfer 12 (3) (1969), 279-300.
[13] E. J. Caron, K. J. Daun, M. A. Wells, Experimental heat transfer coefficient measurements during hot forming die quenching of boron steel at high temperatures. International Journal of Heat and Mass Transfer 71 (2014), 396-404.
[14] R. Watson, in: W. Sharpe (Ed.) Springer Handbook of Experimental Solid Mechanics, 1st Edition. Springer, Ch. Bonded Electrical Resistance Strain Gages, Springer, 2008, pp. 297-302.
[15] R. B. Hetnarski, M.R. Eslami, 2009. Thermal stresses: advanced theory and applications, first ed., Springer, (2009).