A SIMPLIFIED FORGING SIMULATION TOOL: VALIDATION WITH FINITE ELEMENT METHOD

Bachelor Degree Project in Mechanical Engineering
Level ECTS 22.5
Spring term 2013

Omar Zini

Supervisor: Thomas Carlberger
Examiner: Anders Biel
Abstract

Nowadays, the forging industry has a great importance in the fabrication of metallic parts. Therefore, new theoretical models, or modifications to others already existing, continuously appear to improve the process. New investigations culminated, recently, in equations designed to obtain the necessary pressure to forge a part. The purpose is to have a fast, not expensive and efficient alternative to the numerical methods in such calculation.

With the double intention of divulging this new theory created in the University of Málaga (Spain) and confirming its validity, it is interesting to carry out a comparison with the finite element method. In this comparison, geometrical factors relative to the workpiece (part subjected to forging) and the friction coefficient come into play. The goal is then to find validity ranges in the equations for the variables previously mentioned, with the aim to delimit those forging situations in which they could be applied. The implementation of these variables into the commercial FE-code Abaqus is simple, although two aspects arise: the meshing and the correct modeling of the contact between the workpiece and the upper die. The limitation of the Abaqus Student Version (used throughout the analyses) in the allowed number of nodes, prevented performing sufficiently dense meshes. Many alternatives are investigated that, unfortunately, do not show any satisfactory result for the simulation of the process. Performing forging simulations is quite complex and requires a great knowledge in the field and an extraordinary knowledge of how different elements work in large plastic strain.

However, this project leaves a collecting data method and a procedure to determine the initial geometry of the workpiece to ensure plastification and to reach a determined shape factor after the forging. The most important is that the way to further analyses has been cleared by confirming the suitability for certain parameters in the simulation (element type, non-linear geometry, contact model, etc.). The recommendation for future simulation efforts is to use ALE (Arbitrary Lagrangian-Eulerian) adaptive meshing.
Acknowledgements

Through these lines I would deeply like to thank the support offered by the University of Skövde in order to make the present Final Year Project possible. Communication has always been fluid, useful and friendly with each one of the members of the institution with whom I had the pleasure to get in touch.

I would like to make special mention to my supervisor, Thomas Carlberger, whose guidance was essential to overcome the main setbacks arose during the realization of the Project. Kindness, quickness and usefulness are the words to describe his continuous support.

I cannot forget the encouragement of my family and friends to resume this Final Year Project.
Table of contents

Figures and tables ................................................................................................. ivy
1 Introduction .......................................................................................................... 1
   1.2 Background .................................................................................................... 3
2. Structure of the report ....................................................................................... 5
3. The equations ...................................................................................................... 5
   3.1 Metal forming: forging .............................................................................. 5
   3.2 Theory of the equation to test .................................................................. 7
   3.3 The equations (Improvement) ................................................................... 12
4. Technological application of the equations in the forging ......................... 14
   4.1 Machinery .................................................................................................. 15
   4.2 PLC ............................................................................................................. 15
   4.3 FEM ............................................................................................................ 19
   4.4 Target: forging companies ....................................................................... 20
5. Method and results ............................................................................................ 23
   5.1 Preliminary considerations ....................................................................... 23
   5.2 Collecting data (calculations and arrangement) ...................................... 25
      5.2.1 Introduction ......................................................................................... 25
      5.2.2 Design of the forge-workpiece model ............................................. 28
      5.2.3 The mesh: choosing the element type and building the mesh .......... 33
      5.2.4 Material .............................................................................................. 38
      5.2.5 Contact ............................................................................................... 39
      5.2.6 Translation of the process to Abaqus ............................................. 40
   5.3 Evaluation of the results in Abaqus and solutions .................................. 41
6. Conclusions and discussion .......................................................................... 50
7. Future work ....................................................................................................... 52
Bibliography and references ............................................................................... 53
Figures and tables

Figure 1: Open-die forge. On the left, before the deformation. On the right, after the deformation (the free edges of the part are slightly curved) ................................................................. 6

Figure 2: Impression-die forging (Arcabrasives, 2013). ........................................................................ 6

Figure 3: Closed-die forging (Nedians, 2013). ...................................................................................... 7

Figure 4: A double symmetry workpiece to forge and its quarter under study. ......................... 8

Figure 5: A single module divided into three TRBs. ............................................................................. 10

Figure 6: Type of modules according to their position with respect to the vertical symmetry axis. ...................................................................................................................... 10

Figure 7: Values of the angles and corresponding shape of the modules. ............................................. 11

Figure 8: Topology of a module and its kinematics .............................................................................. 11

Figure 9: On the left, the individual P/2k for each module depending on the nature of the friction and including the shape factor, F. On the right, P/2k for the whole workpiece .......... 13

Figure 10: Geometry of a random modular division in the workpiece used as example ............. 13

Figure 11: On the left, an alternative modular distribution. On the right a forbidden distribution (a module always comprises a constant slope angle, α) .............................................................. 14

Figure 12: A PLC with its different components (Wikipedia, 2013) ................................................ 16

Figure 13: On the left, a simple programming device. On the right, a computer (Instrumentación y Control, 2013) ........................................................ .......................................................... 17

Figure 14: On the left, a hydraulic press controlled with a PLC. On the right, keyboard and screen to program the PLC (Toledo Integrated Systems, 2013) ................................................... 18

Figure 15: Geometrical dispositions to avoid in positive angles (α > 0°). On the left, the angle leads to a part impossible to forge (F = 1/tanα). On the right, an unrealistic workpiece is depicted (F > 1/tanα) .................................................................................................................. 26

Figure 16: Disposition of Rₓ, Rᵧ and the force responsible for the forging (F) according to α . . 28

Figure 17: Design of the upper-die ........................................................................................................ 29

Figure 18: Design of the workpiece ...................................................................................................... 32

Figure 19: Assembly of the workpiece and the upper die in Abaqus ................................................. 32

Figure 20: Hourglassing. The reduced and linear element is subjected to pure bending. .......... 36

Figure 21: 4x10 mesh distribution in a quarter. ..................................................................................... 37

Figure 22: a) Ideal plastic solid. b) Strain hardened solid .................................................................... 39

Figure 23: Chart displaying the convergence in Rᵧ (Rᵧ vs number of elements) ......................... 42

Figure 24: Non-convergence in Rₓ vs number of elements ............................................................... 43

Figure 25: Rₓ convergence vs number of elements ............................................................................. 43

Figure 26: Rₓ non-convergence vs number of elements ...................................................................... 43

Figure 27: Workpiece after forging not showing a total plastification ............................................. 44

Figure 28: The workpiece violating the contact at the upper die ...................................................... 45
Figure 29: The undeformed workpiece (grey) vs after deformation (green). The displacement of the bottom right corner is too small. ................................................................. 45

Figure 30: The workpiece meshed with CPE3 elements. Total plastification achieved and violation of the contact of the upper die avoided. ............................................................ 46

Figure 31: Significant displacement of the bottom right corner. ........................................ 47

Figure 32: Two different dispositions of the right side corners after forging. ...................... 48

Figure 33: Workpiece after forging having activated Nlgeom. ........................................... 48

Figure 34: Workpiece exceeding the upper die with Nlgeom = Off. .................................. 49

Table 1. Shape factors, F, to analyze in Abaqus for the comparison........................................ 26

Table 2. P/2k having fixed μ and modified F and α.............................................................. 27

Table 3. Meshes for the convergence study............................................................................. 38

Table 4. Workpiece analysis and values of Rₓ and Rᵧ for different meshing......................... 42
1 Introduction

This Final Year Project aims to validate the equations designed by Martín (2009) to calculate the pressure required to forge metallic parts according to the theories based on the Upper Limit Theorem by Prager (1959) and the Triangular Rigid Block model (Vela, 1983) (Rubio, 2004a, 2004b) (Martín, 2007). These equations intend to be a simple alternative in the calculation of the forging pressure.

The advantage of using such equations will be speed, convenience and simplicity. They will save time and money in the case they are implemented in a simple software with minimal computational costs. The interest is evident in the field of the forging industry where the process could be computerized and controlled by a program that would incorporate the equations. Apart from commercial interests it may also attract researchers in the field who would learn these new equations, study them in depth and make improvements, whenever possible.

The validation procedure will consist in the comparison of the results shown by the equations — for a certain material, friction coefficient and geometry of the part to forge — to those obtained by the finite element method, applied through the Abaqus software. As reproducing the experiments in a real forge is economically infeasible, the values obtained with the finite element method implemented in Abaqus will be considered as the actual ones. Therefore, a criterion by which discard or accept the equations, according to the deviation between their results and those provided by the finite element method, must be set.

If the equations are confirmed to be suitable for a forging process, then a powerful tool, able to replace other procedures more sophisticated but costly from a monetary and temporal point of view, will be available. Otherwise, new courses of action in this field could be provided as a conclusion to the study, so that researchers have solid starting points from which to derive new alternative and more accurate equations. Consequently, the objectives in this Project will be satisfied as long as accurate results for the finite element procedure are achieved in Abaqus, in order to compare them to those of the equations and to reach satisfactory conclusions.
The equations only require the measures of one fourth of the part to forge (workpiece), provided that it shows double symmetry, and the conditions in the contact between it and the upper die of the forge, or, in other words, the type and value of the friction coefficient. Equations (1) and (2) (they will be explained in depth in chapter 3) are to be used depending on the nature of the friction:

- Adhesive conditions (m):

\[
P = \frac{1}{2k} \left( \frac{F_1 \tan \alpha_2}{1 - F_1 \tan \alpha_1} \right) \left( \frac{F_2}{4(1 - F_2 \tan \alpha_2)} + \frac{1}{F_2} \right) + \frac{m}{2} \frac{F_1}{(1 - F_1 \tan \alpha_1)} \left( \frac{F_2}{2 - F_2 \tan \alpha_2} \right) \quad (1)
\]

- Sliding conditions (\(\mu\)):

\[
P = \frac{1}{2k} \left( \frac{F_1 \tan \alpha_2}{1 - F_1 \tan \alpha_1} \right) \left( \frac{F_2}{4(1 - F_2 \tan \alpha_2)} + \frac{1}{F_2} \right) \left( \frac{F_2 (1 - F_2 \tan \alpha_1) + 2F_2}{\cos^2 \alpha_2 (1 - F_2 \tan \alpha_1)(2 - F_2 \tan \alpha_2)} \right) \mu \quad (2)
\]

In the beginning, only the equation for the adhesive friction (m), Eq. (1), was going to be considered, since the one for the kinetic friction coefficient (\(\mu\)), Eq. (2), failed for some geometries (negative values were obtained). Furthermore, due to the usual temperatures and lubrication, forging is usually carried out under adhesive conditions. But, finally, as it will be explained later in chapter 4, Eq. (2) is the equation under study.

Basically, the stages of the present Final Year Project will include, besides the presentation of the theories from which Eqs. (1, 2) come from:

- A method to determine the geometries of the workpiece and friction coefficients for which the values of pressure are to be looked for in Abaqus and Eq. (2).

- Meshing of the workpieces, taking into account that an Abaqus Student Version is used and the number of nodes in the mesh is limited. Those mesh elements more likely to optimize the results will be studied.

- Convergence of the Abaqus results to validate them and, subsequently, use them as the reference in the comparison to the results provided by Eq. (2).
- Tabulation of the results for both procedures (FEM and Eq. (2)) as a function of three different variables relative to the geometry of the workpiece to forge (shape factor and angle) and the technological conditions of the forging (friction coefficient).

- The specification of a comparative approach that should be realistic and effective in validating the results. This is the most challenging part of the project.

- A final stage where to draw conclusions regarding the results.

1.2 Background

Metal forming by plastic deformation processes may be used for the production of parts of any size and shape. They have the advantage of improving the mechanical properties and refining the internal structure of the material to deform. Therefore, they are widely used in the industry nowadays. Among these processes, the forging stands out, where the material is deformed by direct compression.

Since the deformations are permanent, information such as the yield stress and the strain for determined stresses, is needed. However, the friction in the part-tool interface is what conditions the development of the process. Because of the importance of forming processes by plastic deformation, between the 60's and the 90's of the twentieth century there was a development of sophisticated mathematical analysis that resulted in models able to predict the behavior of materials under different conditions (Martín, 2009).

Theoretical models are divided into two families: the numerical and the analytical methods. The latter have been remarkably effective, but as the part gets more complex their effectiveness decreases and, ultimately, become useless. That is why the numerical methods, favored in recent years by the computer development, have taken great importance in the calculations associated with forming processes by plastic deformation. They can analyze any part, no matter how complex it is. However, the disadvantage is their high computational cost.

Martín (2009) developed an analytical method alternative to the numerical methods able to model any part, irrespective of its complexity. The goal was to just modify an already existing analytical method, the Upper Limit Method, in combination with the Triangular
Rigid Block model. The innovation of these investigations lies in the application to any part contour. Moreover, the theory further derived allows quantifying the contribution of effects such as friction, depending on the value and type, temperature (which will not be analyzed in this project), the type of material and even the effects of strain hardening.

However, this new theory introduces a number of restrictions and idealizations that should be taken into account:

- Just parts with double symmetry, leading this fact to study just a quarter of them.

- Plane strain: two dimensional study of the part.

- Absence of inertial and gravitational forces
2. Structure of the report

This report is divided, basically, into three parts:

- The first part is a summary of the technological scope of Eq. (2) to be verified and the theory it comes from. It is important to explain the theory, since it is a novel research that has not reached an important diffusion yet, in order to bring it to the attention of researchers to whom this Project is, mainly, addressed.

- This section is followed by one referring to the method used when performing the experiments. Here it is explained how the technological features of the forge are translated to Abaqus in order to simulate the problem. Guidelines to carry out the simulation are established, as well as a structured method for measuring and arranging the data for their comparison. All the setbacks arisen are reflected with their respective corrections.

- Finally, the conclusions are exposed.

3. The equations

3.1 Metal forming: forging

Metal forming by plastic deformation is the set of processes and techniques designed to permanently change the shape and dimensions of metal parts. Many of these processes are based on direct compression, in which compressive stresses are primarily responsible for the deformation of the material. The list of such processes comprises that of forging, extrusion and lamination, basically. The main list of forming processes is completed by stretching, bending, and deep drawing, among others.

When forging, the material is deformed by direct compression between two dies using impact loads (drop forging) or providing a gradual pressure (press forging). This method is currently used in manufacturing high strength components and in the steel industry (and other metals) to produce large components, requiring a minimum final machining for giving the final dimensions to the workpiece.

A very important factor in the forging processes is the temperature. In fact, most forging is performed in hot conditions, above or below the recrystallization temperature. This
will increase the ductility of the material during the process and reduce the forging force required. On the other hand, a cold working process intends to increase the strength of the finished product by means of strain hardening.

Forging types can be classified according to the manner in which the dies restrict the flow (Groover, 2004):

- Open-die forging → It consists of two plane dies allowing the material to flow without restriction, as represented in Figure 1. The part, generally cylindrical, reduces its height and increases its diameter.

![Figure 1: Open-die forge. On the left, before the deformation. On the right, after the deformation (the free edges of the part are slightly curved)](image)

- Impression-die forging → In this case, the two dies, the lower and the upper, determine the final shape of the workpiece. Initially, the material undergoes a partial compression and then flows out of the dies cavity forming a thin metal layer, called burr. The burr must be removed to obtain the designed dimensions. Figure 2 schematically represents the evolution of the workpiece in this type of forging.

![Figure 2: Impression-die forging (Arcabrasives, 2013).](image)
Closed-die forging → No burr is produced because the material remains within the cavity formed between the punch and the die (Figure 3). The requirements on this kind of forging are superior to the others, because the amount of material must not generate great values of pressure that could damage the die and be sufficient for filling the cavity between the punch and die.

![Figure 3: Closed-die forging (Nedians, 2013).](image)

### 3.2 Theory of the equation to test

The equations to verify are designed to, indirectly, provide the pressure required to forge a metallic part (workpiece). In the work to obtain them, Martín (2009) used a theoretical model, along with certain simplifications. Since the goal was the creation of simple equations as an alternative to the numerical methods, an analytical model was selected as the basis from which to create them. This model is the Upper Limit Method. The Upper Limit Method is based on the Upper Limit Theorem by Prager (1959), oriented to provide the power that is equal to or greater than the minimum necessary to achieve the forging, thus, ensuring the development of the process. Applying great powers would make the forging possible too, however, they are not convenient when they easily exceed the minimum power required, since the process should be developed using least complex and expensive machinery as possible.

The Upper Limit Theorem was applied according to the geometric-kinematic approach proposed by the Triangular Rigid Block model (TRB), developed by Vela (1983), Rubio (2004a, 2004b), Martín, (2007). This model assumes the part to be divided into a specific number of triangular blocks, with the behavior of rigid solids. Consequently, all the points of a rigid block will have the same velocity (absolute velocity) which will be essential for the Velocity Hodograph construction. The Velocity Hodograph shows the kinematic relations (absolute and relative velocities) between triangular blocks and
permits to establish the geometric relations that will be contained in the definitive equations.

The TRB model increases the analytical capabilities and accuracy of the Upper Limit Theorem, with the inclusion of factors such as friction, strain hardening and temperature. It also allows geometric modifications (variation in the number of triangular blocks) towards the optimal solution (that one as close as possible to the minimum power) when calculating the power in a forging process. Other advantages shown by the combination of the TRB model and the Upper Limit Theorem are the distinction between adhesive and kinetic friction, on one hand, and to provide a power value which will always be above the minimum theoretically required, on the other.

The study of the part (from now on, the workpiece) will be two-dimensional, as plane strain is considered, being the deformation in the Z-axis direction negligible. Its field of application affects only workpieces with double symmetry, reducing them to a quarter, to which are imposed the appropriate boundary conditions (Figure 4).

The model is simplified by considering the absence of inertial – quasi-static deformation - and gravitational forces.

The Upper Limit Theorem states:

*The amount of work developed by the surface tensile forces (compression) on an ideal-plastic body is less than or equal to the one carried by the surface tensile forces (compression) for any other velocity field kinematically admissible.*

Considering plane strain and that the workpiece is composed of blocks virtually rigid, Johnson (1959) suggested a new equation according to the Upper Limit Theorem.
Where,

- \( T_i \) [Pa] represents the surface tensile forces applied to the workpiece during its forging.
- \( v_i \) [m/s] is the real field of velocities.
- \( S_v \) [m²] surface on which forces are applied.
- \( k \) [Pa] is the shear yield strength.
- \( v^* \) [m/s] is the admissible virtual velocity field.
- \( S_D \) [m²] is the surface of the boundaries in the triangular rigid bodies (TRB).

The left term of the equation above corresponds to the first part of the statement for the Upper Limit Theorem, representing the minimum actual power to cause and develop the deformation when forging. In turn, the right term is derived from the TRB modeling, where the velocity field is determined according to the division, in triangular blocks, made in the workpiece. The yield strength, \( k \), will be considered in the boundaries of the blocks, where it is necessary to have the shear deformation so that the blocks can have relative motion. The imposition of a virtual field of velocities results from the fact that it is impossible to define the actual field, which is unknown.

Although it was stated that the workpiece was divided into triangular rigid blocks in the TRB model, no distribution has been mentioned so far. It is necessary to determine one to be able to solve the matter about the surfaces and the virtual velocities in the preceding equation. Among all the possible divisions of the workpiece the one that provided the best results was the modular approach.

As its name suggests, the modular approach divides the workpiece into modules, composed each one, in turn, of three TRBs. Each module has three main parameters: its width, \( b \), its height, \( h \), and its slope angle, \( \alpha \). The only warning to be made is that the same module cannot comprise two zones with different slope angles. The division of the material into modules must be such that the chosen configuration provides the closest value to the actual minimum power, by means of the Upper Limit Method and the use of the TRB model. The lowest the value calculated with the distribution, the better the result is. The arrangement of the TRBs in each module always follows the same pattern:
In Figure 5, the three triangles join in the middle point of the bottom side of the module, coincident with the horizontal symmetry axis. The division will be always performed this way. There was a different disposition but it was rejected as its results were less accurate than the actual distribution.

The difference between 'modules with prior module' and 'modules without prior module' must be made, since the mathematical treatment is different in each case. Both types of modules can be identified in Figure 6. In the first ones there is a material inlet velocity to the module, while in the second ones it is nil – module next to the vertical boundary conditions –. Therefore the first module, starting from the left, is always a module without prior module, the rest are all of them modules with prior module.

Besides being classified according to their position, modules must be distinguished according to their angle, $\alpha \neq 0^\circ$ and $\alpha = 0^\circ$ (Figure 7). This was one of the outstanding contributions of the investigations for the equations, the possibility to incorporate angles different from zero, which was never done before. Although in reality it is much more complex, the modules are considered to be affected by a single type of friction (adhesive or sliding conditions) which will keep its value constant throughout the whole process of forging.
Now that the geometry of the module has been defined, in accordance with its angles, it is possible to make up the graph that relates the absolute and relative velocities between TRBs (Velocity Hodograph). This step is necessary to replace the values of the velocities in the equation derived from the Upper Limit Theorem. Despite having referred to the power required for forging, the final equations are meant to calculate $P/2k$, being $P$ (MPa) the applied forging pressure and $k$ the shear strength (MPa) resulting in a dimensionless quotient.

Figure 8 shows an example of a module (without prior module) accompanied by its hodograph:

$V_e$ is the descent rate of the forge. $V_e$ is the speed of the material entering the module. $V_1, V_2$ and $V_3$ are the absolute velocities of the triangular blocks, whereas $V_{12}$ and $V_{13}$ are the relative velocities. Due to the boundary conditions of a module without prior module, $V_e = 0$. The use of the geometric relationships allows both absolute and relative velocities to be linked to each other, solving the inequality and having $P/2k$ as a function of the geometry and the technological conditions during the forging (friction coefficient). Finally, after many calculations these were the two equations, depending on the nature of the friction between the upper die and the upper surface of the workpiece:

- Adhesive conditions,
- Sliding conditions,
\[
\frac{P}{2k} = \frac{1}{2b_2V_1} \left[ \left( \frac{x_2^2 + h_2^2}{h_2} \right) + \frac{(b_2 - x_2)^2}{h_3} \left( V_1 + \frac{(V_e + V_1 \tan \theta_1)}{\cos \alpha_2 - \sin \alpha_2 \tan \theta_2} \cdot \frac{m b_2}{\cos \alpha_2} \cdot \frac{(V_e + V_1 \tan \theta_1)}{\cos \alpha_2 - \sin \alpha_2 \tan \theta_2} \right) \right] \quad (4)
\]

\[
\frac{P}{2k} = \frac{1}{2b_2} \left[ \left( \frac{x_2^2 + h_2^2}{h_2} \right) + \frac{(b_2 - x_2)^2 + h_3^2}{h_3} \right] \left( V_1 + \frac{(V_e + V_1 \tan \theta_1)}{\cos \alpha_2 - \sin \alpha_2 \tan \theta_2} \cdot \frac{\mu}{\cos \alpha_2} \right) \quad (5)
\]

The meaning of every variable can be seen in Figure 8. \( V_e \) and \( V_1 \) are related by the next equation:
\[
V_e = V_1 \frac{\tan \theta_1 + \tan \phi_1}{1 - \tan \alpha \tan \theta_1} \quad (6)
\]

The angles with the subscript ‘1’ refer to those of the module prior to the module under study (provided that it exists). \( V_e \neq 0 \) in case there is a prior module. Since the velocities were virtual in the Upper Limit Theorem, it is possible to give \( V_1 \) any value. Usually, \( V_1 = 1 \text{m/s} \).

With respect to friction, the range in \( m \) is \([0, 1]\) and in \( \mu \) \([0, 0.577]\).

If the equations are to be applied to the workpiece, divided into several modules, then one must calculate the individual values of \( P/2k \) in every module and, subsequently, combine them to obtain the global \( P/2k \). To do so, the next equation is used:
\[
\frac{P}{2k} = \frac{1}{2k} \frac{1}{\sum_i b_i} \sum_i P_i b_i \quad (7)
\]

### 3.3 The equations (Improvement)

The equations were improved by Zini (2010) replacing some geometrical parameters by the so called shape factor \( F \), taking into account the relationship between \( V_e \) and \( V_1 \) and removing all the angles except \( \alpha \). The shape factor is the quotient resulting from the division of the width of the module, \( b \), and its height, \( h \): \( F = b/h \). It is proved that workpieces with the same shape factor (being equal the rest of the parameters) lead to
the same results of $P/2k$. Figure 9 shows the morphology of the new equations, used in this FYP.

\[
\frac{P}{2k} = \left(1 + \frac{F_1 \tan \alpha_2}{(1 - F_1 \tan \alpha_1)} \right) \left(\frac{F_2}{4(1 - F_2 \tan \alpha_2)} + \frac{1}{F_2} \right) + \frac{m}{2 \cos^2 \alpha_2 \left(2 - F_2 \tan \alpha_2\right)} \left(F_2 + \frac{2F_1}{1 - F_1 \tan \alpha_1}\right)
\]

Adhesive Conditions

\[
P = \frac{1}{2k} \left(\frac{F_2}{4(1 - F_2 \tan \alpha_2)} + \frac{1}{F_2} \right) \left(1 - \frac{F_2 \left(1 - F_1 \tan \alpha_1\right) + 2F_1}{\cos^2 \alpha_2 \left(1 - F_1 \tan \alpha_1\right) \left(2 - F_2 \tan \alpha_2\right)} \right) \mu
\]

Sliding Conditions

Global formula

Figure 9: On the left, the individual $P/2k$ for each module depending on the nature of the friction and including the shape factor, $F$. On the right, $P/2k$ for the whole workpiece.

The subscript ‘2’ is for the module under study and ‘1’ for the prior module. When there is no prior module, then:

\[
\frac{F_1}{1 - F_1 \tan \alpha_1} = 0 \quad (8)
\]

A quarter of a workpiece has been considered in Figure 10 as an example of application of the equations:

Figure 10: Geometry of a random modular division in the workpiece used as example.

- First module starting from the left (module without prior module):
  - $b_2 = 50$ mm, $h_2 = 20$ mm; $\alpha_2 = 5^\circ$; Adhesive conditions ($m = 0.9$)
\[ F_2 = \frac{b_2}{h_2} = 2.5; \quad F_1/(1-F_1\tan\alpha_1) = 0 \rightarrow P/2k = 1.84 \]

- Second module starting from the left (module with prior module):

\[ b_1 = 50 \text{ mm}, \quad h_1 = 20 \text{ mm}, \quad \alpha_1 = 5^\circ; \quad b_2 = 25 \text{ mm}; \quad h_2 = h_1-b_1\tan\alpha_1 = 15.63 \text{ mm}; \quad \alpha_2 = 0^\circ; \quad m = 0.9 \]

\[ F_1 = \frac{b_1}{h_1} = 2.5; \quad F_2 = \frac{b_2}{h_2} = 1.6 \rightarrow P/2k = 2.63 \]

Consequently, the global value is:

\[
\frac{P}{2k} = \frac{1}{2k} \sum_{1}^{i} P_i b_i = \frac{1.84 \cdot 50 + 2.63 \cdot 25}{50 + 25} \approx 2.10
\]

It is important to remember that there are infinite possibilities to arrange the modules for a determined workpiece and, therefore, infinite values of P/2k among which the smallest will be the best one. Figure 11 is referred to the allowed and forbidden distributions inside a quarter.

4. Technological application of the equations in the forging

This is a summary about hammers and presses used in the forging industry, the features of the electronic controllers that govern them and the use of FEM based software in the forging companies. The idea is to show the best format in which to implement the software based on Eq. (1) and Eq. (2) and to find out the most likely type of forging companies that could be interested in it.
4.1 Machinery

The machinery used is classified (Groover, 2004) based on how it impacts the delivered load against the material to be forged. On one side there are forging hammers, characterized by providing the load with impacts. On the other side, forging presses (or just presses) which provide load gradually and continuously.

Forging hammers are used in impression-die forging, where the upper die is fixed to the ram and the lower to the anvil, which acts as a support. The ram is raised and then dropped, crashing into the working material - repeating the operation several times -. In turn, forging hammers are divided into those based on free fall gravity and the power supplied by a fluid. In the first type, the impact energy is determined by the height from which the ram falls and its weight. In the second, the movement of the ram is governed by the action of air pressure or steam.

Forging presses are subdivided into mechanical, hydraulic and vise presses. In the mechanical ones, a series of mechanical elements transform the engine rotational movement into the translation of the punch. In the second ones, the punch is activated by a hydraulic cylinder, whereas in the vise presses one vertical spindle moves the punch.

Nowadays, the forging machinery is automated for reasons lying on the security and the economic and productive efficiency. Initially, its automation fell on relays that, later, were replaced by PLCs. Such devices use software that, apart from processing the external signals, allow the worker to give instructions for the process control. Is in the PLC where the implementation of the software based on the forging-load equations has to take place.

4.2 PLC

A PLC (Programmable Logic Controller) is a device responsible for the control of electromechanical industrial processes. In the forging industry it is used to govern the machinery operation (presses and hammers). A PLC is composed of a central processing unit (CPU), the Input/Output (I/O) interfaces and the power supply.
The I/O interfaces receive the signals from the sensors and, according to them, the CPU runs the corresponding program. From the results generated by the program after the data processing, the CPU orders the output interface to connect or disconnect certain actuators (electrical or mechanical devices). The CPU consists of the memory, the processing system and the circuits. PLCs can be accompanied by, besides the elements named before, a programming device (which it does not belong to the PLC itself) and one or more memory modules, not to be confused with the CPU memory. These two components are going to be discussed below.

There are different memory formats (Tokheim, 1987) in which to store the software derived from the codification of the equations for the forging load (Martín, 2009) (Zini, 2010). Such memories are connected to the PLC, being the following the most important types:

- **RAM memory** → In this kind of memories it is possible to read and to rewrite the information. It is a volatile memory, which means that if there is a power cut, then all the information, including the software, is erased. To avoid this drawback a standby battery source is used, lasting from two to seven years.

- **EPROM memory** → Unlike Ram memories this type is a non-volatile memory. They are commonly used to store the software with which the worker interacts, once that it has been conveniently refined and proved to run without errors. To erase all the information they contain, in order to write a new program, it is necessary to use an ultraviolet light.
- EEPROM memory → Another non-volatile memory. It has the same features as the preceding type of memories, except for the way they are erased: in this case, by electrical means.

- RAM + EEPROM → Not a type of memories but the most common way to combine them. If a power cut occurs, then the information (and, therefore, the software) is transferred onto the EEPROM memory. Once the electricity is restored the information can be sent back to the RAM memory again.

With respect to the programming device, it allows the worker to read, write, modify and monitor the PLC. The PLC programming can be carried out by means of simple devices similar to a calculator (Figure 13a) or with the help of a keyboard and a screen, like in Figure 13b. In both cases, a communication wire connecting the PLC to be programmed and the device is required. The programming device is independent of the PLC.

![Figure 13: On the left, a simple programming device. On the right, a computer](Instrumentación y Control, 2013)

When purchasing/installing a PLC some factors have to be taken into account: the memory of the CPU, the processor and the number of I/O interfaces. All of them have an effect on the cost of the PLC.

But which would be the features and benefits of encoding a program with the equations for the forging load on a PLC?

1º) The programming would be easy as implementing two equations, the fields for the geometry of the modules and their friction coefficient and the properties of the material.

2º) The software, very light, would not raise the price of the PLC, since it would not need a powerful processor nor a CPU memory expansion. A RAM+EEPROM memory,
which is easily attachable to a modular PLC, would house the software. In conclusion, the only is the RAM+EEPROM itself and the programming.

3º) The CPU always gives priority to software in a module against the content of the internal memory (RAM in the CPU) of the PLC, so that attaching the RAM+EEPROM to the PLC would be sufficient to run the software.

4º) The PLC handling, after the software installation, would not require an additional training for the workers, due to its simplicity. The interaction between the worker and the forging process, through the software, would be made with the keyboard and the screen (Figure 14) writing the inputs such as the ‘height’ of the quarter (measured on the vertical symmetry axis) the ‘width’ of the modules and their respective ‘angle’, the ‘material’ (Yield stress of the metal) the ‘friction coefficient’ and the presence or not of strain hardening.

![Figure 14: On the left, a hydraulic press controlled with a PLC. On the right, keyboard and screen to program the PLC (Toledo Integrated Systems, 2013)](image)

5º) Saving money in FEM software license.

6º) Saving time in the analyses.

The software, under the RAM+EEPROM format, could be integrated by the sellers of hydraulic presses (Loire Safe, 2013) controlled by PLCs, for instance. This would be an incentive for the purchasing forging companies with the intention of being able to do without the FEM analysis, with respect to the forging load. It also could be offered by those companies responsible for updating obsolete systems, in both aspects, mechanical and electronic. As an example, Toledo Integrated Systems (Toledo Integrated Systems, 2013) is an American company who offers technical solutions in those fields.
The software would include its corresponding license and only the approved supplier would have access to the code, in order to modify it according to the changing conditions of the forging process. This software is usually provided with a key that prevents non authorized personnel to rewrite the lines of the code. At the same time, with this step the possibility of an illegal distribution and use of the software is avoided.

4.3 FEM

In this section, all the possible solutions that a FEM based software can provide to the forging industry and the necessary inputs to carry out the corresponding calculations will be explained. The intention is to show the technological scope of the FEM to, subsequently, position its use in a realistic context since due to economic and productive reasons not all the forging companies own a FEM based software or all its features. In fact, there are two situations in the case of forging companies:

- Those which technical department does not perform FEM calculations because the production design, very standardized, is based on experience.

- Those in possession of a license of a FEM based software. This usually happens in important companies in position to offer a very versatile production, easily adaptable to almost any shape – there is a restriction in the weight of the workpiece, though –.

Inside the last type of companies, there are different situations with respect to the sophistication of the FEM based software used. Licenses do not have a fixed price, it depends on the modules/complements hired. To conclude, the target is to find under which technological and economic circumstances the software based on the equations (Eq. (1), Eq. (2)) can be proved to show more advantages than that of the FEM.

The implementation of the FEM in the forging field pursues the following goals (Euskal Erriko Unibertsitatea, 2013):

- Determination of the load and the energy to forge the workpiece.

- Simulation of the preforms behavior, which are the different form-phases of the metal from the beginning to the final design.

- Analysis of the material flow.

- Influence of the temperature and its gradient.
- Determination of the stresses and deformations in the dies.

To perform the analysis with the FEM software the initial data, below, are essential:

- A CAD file with the initial geometry of the metal and that of the dies.

- Equations relative to the elastoplastic behavior of the material. Stresses, deformations, deformation rates and temperatures are related.

- Material of the dies. The die is supposed to work under elastic conditions.

- Lubricant properties.

- Machinery (hammer or press) features.

4.4 Target: forging companies

Two forging companies, Forging Products and Alcorta Group, were asked to answer some questions with respect to their working methodology, to know, thus, the procedures of their own technical departments in relation to the analyses of the forging parameters. The goal of these interviews was to understand the different ways each forging company could carry out its designs, to see its needs and, consequently, to study into which ones the forging-load software would fit better. It can be considered as a rudimentary market research.

The first interviewee was the director of the technical department in one of the two factories that Forging Products owns in the North of Spain, situated in Legazpi. Forging Products furnishes components (ranging from 3 to 50 kg) to very important companies within a great variety of sectors – Nissan, Scania, Rexroth, etc. – but it mainly produces elements for the transmission and the steering of industrial vehicles. The sequence of their forging process is summarized as it follows:

1°) The customer has to supply the drawing of the workpiece to be forged, which design is usually based on the FEM. This design has nothing to do with the forging, but with the features of the workpiece related to its functionality.

2°) This factory works with tables that provide the technological parameters of the forging. In such tables, the shape of the contour of the workpiece and the metal it is made of determine simple but important values: the forging load, the stresses in the
dies, the temperatures reached in the process, etc. All the values contained in the tables are based in calculations which accuracy has been endorsed by the experience of working with the same type of workpieces through the years.

3º) At least in the Legazpi factory, the forging is performed with hammers working at an 80% of their maximum capacity (being all of them oversized in a 20%) always above the power that would result from an accurate calculation. The ‘overpowered’ forging is a consequence of the necessity to ensure the process. A greater accuracy in the power supplied would lead to savings, above all, with respect to the wear of mechanical elements, such as bearings, which would be replaced with less frequency. In addition, the temperatures generated could be better controlled.

4º) The PLCs in the factory are used as a very simple alternative to relays. They automate the forging without the worker operating on them (unless diagnosis). Programming devices are not used since the PLCs accomplish simple tasks.

The second interviewee was the director of the technical department in Alcorta Group (Elgoibar, España) a company dedicated to the fabrication of components for the automotive industry. Here, the resources were much more sophisticated than in Forging Products. Once the drawing of the workpiece is sent by the customer to the company, the technical department calculates the load to apply on it, the stresses on the dies, the material flow, the design of the preforms, etc. For all these operations a FEM software is required. All these calculations are very important to avoid technical errors throughout the process such as defective surfaces (cracks) in the workpiece. Both, the company and the customer have to arrive to a technical agreement, therefore, if necessary, the latter will have to slightly modify the design of the workpiece, without altering its original performance, so that it can fit into the machinery features and the manufacturing techniques of the company.

The director of the technical department also provided very valuable information about the price of the license in a FEM based software. Without offering an exact value, just ranges, it was reported that the price of the license depends, basically, on two factors: the software manufacturer and the complements. The license has to be paid for every terminal (computer) the software is installed in, except for special financial arrangement between big companies and the software manufacturers. The most economical licenses
in this kind of software, without complements, move around 5,000-12,000 euros a year. There are licenses even more inexpensive, 3,000-5,000 euros, but in the case of design programs with just some calculation functions. A FEM based software, with the necessary complements for an accurate forging design may range from 20,000 to 80,000 euros a year in an average company. This would be the cost to purchase the software, while the cost for the annual renewal moves between 2,000-10,000 euros. The licenses for very powerful software, with many complements, can easily exceed 100,000 euros.

For a company like Forging Products, where a calculation by the FEM is not used in the forging design, it would be a great chance to update their PLCs and to implement the program (in a RAM+EEPROM memory) responsible for the forging load that would lead to more accurate calculations, resulting in a decline in the wear of the mechanical elements of the machinery and a better control of the temperatures and lubricants. This way, Forging Products will achieve a qualitative leap without having to pay for any FEM based software license that, even if simple, will always represent a significant cost in comparison to the forging-load software. Not to mention the savings in qualified personnel as the program is very intuitive.

In Alcorta Group, the use of FEM analysis is not limited only to the forging loads, but it is used to give solution to many other technical aspects mentioned before. Implementing the software to the PLC would bring time savings for the forging load analysis; however, the technical department still would make use of the FEM software since stresses, temperature gradients, etc. have to be solved. Even more, it is preferable to calculate the forging load within the same analysis the other forging parameters come from, so as to keep in line with them. Besides, the forging-load program is not intended for companies with large productions and high level quality, in which the use of the FEM is irreplaceable because of its accuracy.

So, in short, the forging-load software to be implemented in PLCs is thought to be used in small and medium forging companies that do not incorporate calculations based on the FEM by economic and production accuracy reasons.
5. Method and results

5.1. Preliminary considerations

The aim of this Final Year Project is not the total approval or rejection of the equations derived to, indirectly, provide the pressure to apply in a forging process. The aim is to determine their validity ranges depending on the combination of the variables that characterize them, to understand in which technological cases they can have a real and accurate application.

These variables determine the final value of \( P/2k \) in the equations for a single module (Eqs. (1, 2)):

- The friction coefficient: kinetic \((\mu)\) or adhesive \((m)\).
- The shape factor of the module under study \((F_2)\) and its angle \((\alpha_2)\).
- The shape factor of the prior module \((F_1)\) and its angle \((\alpha_1)\).

Thus, the intention is to determine for which ranges of values for each one of these variables satisfactory results of \( P/2k \) are achieved. Depending on its geometry, the study of a metallic workpiece may require, necessarily, the use of more than one module. This happens when, in the metal contour, there is more than one face in contact with the upper die. In this case the overall pressure (Eq. (7)) to apply for forging is obtained from the individual pressures for each one of its modules.

To check the validity of the theory related to the equations from the mentioned global value of pressure calculated, in turn, as the combination of the individual pressures would be an impossible task to carry out according to the multiple variables when increasing the geometric complexity of the workpiece. Thus, it was thought advisable to study one-module workpieces.

So only the equations for one-module metals (without prior module) are going to be studied. Thus, there is no material entering the module and, as a consequence, the flow/speed of the flow will be equal to zero (Eq. (8)).

Then the variables are now three: the shape factor of the metallic part \((F_2)\), the friction coefficient \((\mu/m)\) and the angle \((\alpha_2)\). From now on, \( F = F_2 \) and \( \alpha = \alpha_2 \).
With respect to the friction coefficient the analysis was intended to be made with adhesive friction due to two reasons:

- The deformation in the forging processes is usually governed by adhesive conditions in the die-metal interface.

- The equation for the kinetic friction ($\mu$) gives negative values of $P/2k$ for certain geometries. It was thought that this did not happen in the equation for the adhesive friction ($m$) but it was found later that, occasionally, it gives negative values too.

As the way to perform an analysis in Abaqus under adhesive conditions was not found, an alternative analysis with $\mu$ was preferred as a first approach, without losing the intention to find a solution to the issue.

It should be remembered that the most reliable method to test the validity of the results would be to have a real forge and the metal to deform. But considering the cost of the machinery, the technical knowledge for its handling, the exorbitant price of the amounts of metal to forge (not to mention their supply), the available time, etc. the situation forces to take a cheaper and closer reference: the finite element method (FEM) implemented in the program Abaqus. In turn, to prove the accuracy of the Abaqus results they will have to satisfy convergence requirements and other technological conditions (total plastification, incompressibility, displacements in determined nodes…) otherwise there will be no sense in considering them the ‘real’ values to compare with. The version of Abaqus used is the Student Edition 6.12-2.

Pressure will be applied gradually (press forging), and not by impacts, so the upper die in the corresponding Abaqus animation will move down at a constant speed. Among all the types of forging, considering the manner in which material flows, the problem has been designed for an open-die forge in which there is no excess material (burr) to be removed after. Furthermore, the burr would imply additional pressures that have not been introduced neither in the equations nor the simulation.

Depending on the nature of the problem simulation, Abaqus can solve it by using the methods associated with implicit (Abaqus/Standard) or explicit procedures (Abaqus Explicit). Basically, Abaqus/Standard is used in static and quasi-static problems, whereas Abaqus/Explicit is for dynamic problems. This means that, having considered the forge as quasi-static when deriving the equations, the analytical product to use is
Abaqus/Standard. On the other hand, Abaqus/Explicit cannot be ruled out in quasi-static problems in which Abaqus/Standard has difficulties in finding the convergence of the values. It can be very useful under contact conditions, such as the ones existing between the forge and the workpiece, where the convergence may turn out to be problematic. For this reason, Abaqus/Standard will be regarded as the analytical product to use but without excluding the possible utilization of Abaqus/Explicit to fix convergence problems.

5.2 Collecting data (calculations and arrangement)

5.2.1 Introduction
The method to perform the comparison between the results of the equations and those with FEM intends to tabulate, first, the \( P/2k \) values obtained by the Abaqus analyses having fixed, step by step, two variables and then modifying the remaining one. For example, the friction coefficient will be the first to be fixed. Then, from among the entire set of angles to be studied, one of them will be chosen and the shape factors of interest will be modified to provide the \( P/2k \) values.

These are the selected representative values to be implemented in Abaqus and in the equations to make the comparison for \( P/2k \):

- Angles, \( \alpha \): \(-60^\circ, -45^\circ, -30^\circ, -15^\circ, -10^\circ, -5^\circ, 0^\circ, 5^\circ, 10^\circ, 15^\circ, 30^\circ, 45^\circ \) and \( 60^\circ \)

- Kinetic friction coefficient, \( \mu \): 0.1, 0.2, 0.3, 0.4 and 0.5 (Maximum = 0.577 according to the von Mises yield criterion in pure shear)

- Shape Factor, \( F \): 1,2,3,4,5,7 and 10 (only for \( 0^\circ \) and negative angles)

By its own definition, there are limitations on the values that the shape factor \( F \) can adopt depending on the angle \( \alpha \). This restriction only exists for positive angles, in which case the use of inappropriate shape factors will lead to inadequate or unrealistic geometries. The limitation for positive angles is determined by the inverse of their tangent. Since \( F = b/h \) the limit situation would be that one in which \( \tan(\alpha) = h/b = 1/F \). Thus, the maximum value of \( F \) for a positive angle, \( \alpha \), coincides with the inverse of the tangent of the angle: \( F_{\text{max}} = 1/\tan(\alpha) \). Figure 15 displays the geometrical issues related to \( F \).
Figure 15: Geometrical dispositions to avoid in positive angles \( (\alpha > 0^\circ) \). On the left, the angle leads to a part impossible to forge \( (F = 1/\tan \alpha) \). On the right, an unrealistic workpiece is depicted \( (F > 1/\tan \alpha) \).

It should be noted that the shape factor to consider will be that of the deformed workpiece and not the one in the beginning of the process, as it will be justified later. Table 1 shows the maximum values of \( F \) for each positive angle and the selected shape factors to study (the shape factors used in the negative values and \( 0^\circ \) coincide with those for \( \alpha = 5^\circ \)):

<table>
<thead>
<tr>
<th>Positive angles</th>
<th>Maximum Shape Factor (F)</th>
<th>Used Shape Factors</th>
</tr>
</thead>
<tbody>
<tr>
<td>5°</td>
<td>11.43</td>
<td>0,1,2,3,4,5,7,10</td>
</tr>
<tr>
<td>10°</td>
<td>5.67</td>
<td>0,1,2,3,4,5</td>
</tr>
<tr>
<td>15°</td>
<td>3.73</td>
<td>0,1,2,3</td>
</tr>
<tr>
<td>30°</td>
<td>1.73</td>
<td>0,0.25,0.5,0.75,1,1.5</td>
</tr>
<tr>
<td>45°</td>
<td>1</td>
<td>0,0.25,0.5,0.75</td>
</tr>
<tr>
<td>60°</td>
<td>0.58</td>
<td>0,0.25,0.5</td>
</tr>
<tr>
<td>0 and Negative angles</td>
<td></td>
<td>0,1,2,3,4,5,7,10</td>
</tr>
</tbody>
</table>

Table 1. Shape factors, \( F \), to analyze in Abaqus for the comparison

Therefore, considering the process to be followed, it would end up having several tables collecting the values of \( P/2k \) for each one of the different friction coefficients, as shown in Table 2. There have been left blank spaces in Table 2 fields in the case a value is expected, while in those fields where there are asterisks it will not be calculated (whether it is because of eventual geometric errors or just because for those angles it was not interesting to study other shape factors). This table would be repeated for the rest of the values of \( \mu \) (0, 0.1, 0.3, 0.4 and 0.5):
Table 2. P/2k having fixed $\mu$ and modified $F$ and $\alpha$

Combining the angles with the friction coefficients and the diverse shape factors, the total number of technological situations to study will be equal to $6 \cdot 87 = 522$, with their respective values of $P/2k$. This is the initial approach, although maybe the huge quantity of individual analysis to carry out and the likely lack of time will imply the omission of those expected to not be representative or reiterative, therefore, useless.

Abaqus does not calculate $P/2k$ because such a dimensionless value has no application to any known technological problem. In fact, although the equations are designed for $P/2k$, its purpose is to provide the user with the necessary pressure, the really implementable parameter to control in the forging process. In turn, $P$ will not be the result shown in Abaqus either, but that of the components of the force exerted by the upper die on the workpiece.

All finite element simulations require further convergence study of the analyzed variables for finer meshes in order to establish the final, more accurate, value to compare with that provided by Eq. (2). The convergence study is to be performed not directly on the forging pressure $P$ but on the horizontal $R_x$ and vertical $R_y$ components of the upper die force applied to the metal to forge (Figure 16). The idea is to check this convergence starting from coarse meshes and moving to finer ones, in which a more accurate value is expected. Once the convergence for $R_x$ and $R_y$ is confirmed, both will be projected on the perpendicular direction to the contact surface, being this the force to divide by the area of the latter to have the forging pressure.

$$\alpha \neq 0^\circ \rightarrow F = -R_y \cdot \cos \alpha - R_z \cdot \sin \alpha \quad (9)$$
\[ \alpha = 0^\circ \rightarrow F = -R_y; R_x = 0 \] (10)

As it is a plane strain problem, the thicknesses for all the workpieces have been determined to be equal to 1 m. Multiplying the length of the deformed contact surface \((b_f / \cos(\alpha))\) by the thickness, where \(b_f\) is the width \((b)\) of the forged workpiece:

\[
A \text{ (Contact Area)} = (b_f / \cos(\alpha)) \cdot \text{thickness} \rightarrow A = b_f / \cos(\alpha) \quad (11)
\]

\[ P = F/A \rightarrow k = \sigma_Y/\sqrt{3} \text{ (von Mises criterion)} \rightarrow P/2k \]

Figure 16:Disposition of \(R_x, R_y\) and the force responsible for the forging \((F)\) according to \(\alpha\).

**5.2.2 Design of the forge-workpiece model**

In Abaqus, a model is the representation of a part or a set of parts to which apply the finite element method. This is the first step in the analysis of any workpiece in Abaqus. In the forging process, the parts in the assembly are two – the upper die and the metal to forge (workpiece) – as a consequence of the geometry of the plane strain problem. Due to the double symmetry of the workpiece, the analysis will focus just in one quarter of it, the upper right quarter, to be more precise. Then, the study is reduced to the interaction between the latter and the upper die. As the deformation occurs under plane strain conditions, both parts (upper die and workpiece) have to be defined in the \(x-y\) plane. Because of the nature of the problem, the upper die shows a greater stiffness than that of the metal to deform. Consequently, it was idealized as a rigid solid that will not suffer any deformation. On the other hand, the workpiece was defined as a deformable material.

The upper die was set as an *analytical rigid* that, in contrast with a *discrete rigid*, does not require any meshing. Despite having the appearance of a simple ‘wire’, its design has to satisfy two basic characteristics to avoid errors:
- The length of its surface (in contact with the upper face of the workpiece) has to be long enough to keep the deformed material enclosed. In other words, in order to prevent the material to flow beyond the forge boundaries. Otherwise, the workpiece would flow totally free introducing unrealistic stresses that would falsify the results. The length has to be, at least, equal to the final length (once the forge is achieved) of the upper contact surface of the workpiece, which can easily be calculated.

Both ends of the upper die are always vertically lengthened 0.1 m to prevent any node of the workpiece upper-surface to be able to ‘fall behind’ the surface of the former. Actually, because of the vertical symmetry axis and the oversizing of the upper die length there should not be any problem in none of its ends regarding this aspect. Even so, with the aim to prevent unexpected events in Abaqus, it has been preferred to design it that way and not leaving anything to chance. The corners have been rounded (fillet, 0.05 m radius) in imitation of other similar examples in Abaqus. The fillets are usually used to reduce the stress concentrations that take place in corners in contact with a rigid body. See this design in Figure 17.

![Figure 17: Design of the upper-die.](image)

To define the workpiece, it is necessary to specify its height $h$, its width $b$ and its angle $\alpha$. In the equations, $F$ is the shape factor of a workpiece plastically deformed that has to undergo more forging deformation (in the plastic region), implying that the data collection is to be performed, exclusively, with geometries already plastified. In the case of Abaqus, however, the workpiece will be, first, elastically deformed to, subsequently, reach the plastic region. That is the reason for setting the first objective to subject the workpiece to a descent of the upper die such that it ensures the total plastification in order to measure, for the deformed geometry, the components of the applied force. As a
consequence, a relation between the shape factor of the deformed workpiece \( (F_f, \text{ final shape factor}) \) and that for its undeformed contour \( (F_i, \text{ initial shape factor}) \) is going to be made including the descent of the upper die. It is vital to find this equation, since the geometry to implement in Abaqus is that of the undeformed metal. \( F_i \) is the value that coincides with \( F \) in the equations.

In every analysis, the initial height \( (h_i) \) of the left side is going to be dimensioned, by default, with 1 m length. Thus, since \( F_i = b_i/h_i \), it is possible to specify the initial geometry for which it is necessary, in addition, to know the vertical displacement of the upper die that will lead to the value of \( F_f \). Theoretically, according to the plane strain idealization taken into account in Eq. (2) and the incompressible behavior of the metal, the area \( A \) of the workpiece in the \( x-y \) plane is constant. This fact allows connecting \( F_i \), \( F_f \) and the vertical displacement of the upper die \( (\text{def}) \). Below the process is described to derive the function \( F_i = f(F_i,\text{def}) \) in three different situations: \( \alpha < 0^\circ \), \( \alpha = 0^\circ \) and \( \alpha > 0^\circ \)

\( \alpha = 0^\circ \)

\[
F_i = \frac{b_i}{h_i} ; \quad h_i = 1 \rightarrow b_i = F_i
\]

\[
F_f = \frac{b_f}{h_f} ; \quad h_f = h_i - \text{def} = 1 - \text{def} \rightarrow b_f = F_f \cdot (1 - \text{def})
\]

\[
A = b_f \cdot h_f = b_i \cdot h_i = b_i = F_i \rightarrow \frac{b_f \cdot h_f}{h_f^2} = \frac{F_i}{h_f^2} \rightarrow F_f = \frac{F_i}{(h_i - \text{def})^2}
\]

\[
\rightarrow F_f = \frac{F_i}{(1 - \text{def})^2} \Rightarrow F_i = F_f (1 - \text{def})^2 \quad (12)
\]

For example, \( h_i = 1 \), \( \alpha = 0^\circ \), \( \text{def} = 0.1 \, \text{m} \), \( F_i = 4 \rightarrow F_i = F_i (1-\text{def})^2 = 4 \cdot (1-0.1)^2 = 3.24 \rightarrow b_i = 3.24 \, \text{m} \). The initial workpiece is 1 m high and 3.24 m width.

\( \alpha > 0^\circ \)

The general equation for the area is:

\[
A = b \cdot h - \frac{b^2 \cdot \tan \alpha}{2} = \frac{2 \cdot b \cdot h - b^2 \cdot \tan \alpha}{2} \quad (13)
\]

Dividing by \( h^2 \):
Moving terms will result in a quadratic equation:

\[ -F_i^2 \tan \alpha + 2F_i - (2F_f - F_f^2 \tan \alpha)(1 - def)^2 = 0 \]  \hfill (14)

Where,

\[ a = -\tan \alpha; \quad b = 2; \quad c = -(2F_f - F_f^2 \tan \alpha)(1 - def)^2 \]

To illustrate its use, a practical situation is going to be solved. The initial data are \( \alpha = 15^\circ, \quad h_i = 1 \text{ m}, \quad F_i = 1, \quad def = 0.1 \text{ m}. \) It only consists in solving the quadratic equation:

\[ F_i = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a} = \frac{-2 \pm \sqrt{2^2 - 4(-\tan 15\degree)(-1)(2 \cdot 1 - 1^2 \tan 15\degree)0.9^2}}{2(-\tan 15\degree)} \]

\[ F_i1 = 0.78; \quad F_i2 = 6.68 \]

The right solution will always be that in which \( F_i < F_f, \) since in the whole process of forging, when the width of the workpiece increases at the same time its height decreases, an increase in the shape factor will always take place. In fact, the value of 6.68 would lead to an unrealistic workpiece geometry divided into two regions joined by a point, similar to a bow tie.

Then \( F_i = 0.78 \) and, therefore, \( h_i (\text{mm}) = 0.78 \text{ mm} \) would be the value to use to build up the workpiece. The construction of the quarter is represented in Figure 18.
The minimum value for the length of the upper die in this example would be: \( F_l = 1; h_l = h_i - \text{def} = 1 - 0.1 = 0.9 \text{ m} \rightarrow b_l = F_l \cdot h_l = 1 \cdot 0.9 = 0.9 \text{ m} \rightarrow \text{Length (Min)} = b_l / \cos(\alpha) = 0.9317 \text{ m}. \) This measure is for the straight surface of the upper die. However, if the fillets are to be taken into account then: \( \text{Length(Min)} = 0.9317 \text{ m} + 2 \cdot 0.05 \text{ m} = 1.0317 \text{ m}. \) The upper die was given a length of 3 m (for the straight region). In this example, the upper die was designed so long because its descent when forging did not compromise the analysis. Anyway, for positive angles there is always a limitation in its maximum length, since at the end of the forging the upper die would crash into the horizontal symmetry axis, originating an analysis error. The arrangement of both, the workpiece and the upper die, would be as in Figure 19.

\[ \alpha < 0^\circ \]

In a negative angle, considering its sign, the equation to calculate the area is the same as that in the case of positive angles and, therefore, the preceding quadratic equation is accepted in the present case.
5.2.3 The mesh: choosing the element type and building the mesh
When meshing a part, a number of essential factors must be incorporated to make the correct selection of the element type. Abaqus/Standard has a wide variety of elements grouped in ‘families’: Continuum/Solid, Shell, Beam, Truss etc. With respect to the forging process, the mesh will be composed of Continuum/Solid elements. The Solid elements family is divided, on one hand, into those three-dimensional (hexahedra, tetrahedra and wedges) and, on the other hand, the two-dimensional elements. Due to the nature of the problem the selection has to be made among the two-dimensional elements with plane strain (PE). Quadrilaterals and triangles are the two existing possibilities in 2D. Usually, the former are more accurate than the latter, although these are still used because of their good adaptability to complex geometries, something really interesting in those workpieces with angles \( \alpha \) such that can give rise to excessive distortions in the rectangular elements.

Elements in Abaqus are identified with an alphanumeric code that informs about their family, dimensionality, number of nodes (direct relation with the shape and interpolation order), integration and formulation. The first letter points to the family. In the case of the Continuum/Solid elements it is a C. Inside the family, the elements destined to plane strain use the acronym PE (those for the plane stress, for example, are identified as PS) that, at the same time, refers to the dimensionality of the element (two-dimensional). This information is followed by the number of nodes (dependent of the interpolation order). After this, a letter referring to the type of integration: just in the case it has a reduced integration an R appears in the name of the element. The formulation refers to the mathematical theory that governs the behavior of the element throughout the analysis. In the case of the Continuum elements, Abaqus uses a lagrangian behavior, by default, to which there is not any identification. However, there are other formulations that are explicitly indicated in the name of the element with a letter (e.g ‘H’).

For example: CPE4RH is a Continuum element used in meshes when there are plane strain (CPE) deformations. It has four nodes, hence, it is rectangular with a linear interpolation. ‘R’ stands for ‘reduced’, the type of integration, and ‘H’ means that the hybrid formulation is used instead of the lagrangian.
When choosing the element type of the mesh, it is vital to know its type of integration and order of interpolation. This is why both terms are going to be revised.

There are two types of elements depending on their interpolation orders: linear or first-order elements and quadratic or second-order elements. In a stress/displacement problem, Abaqus calculates, first, the displacements measured in the element nodes to derive from them the values of other variables (stresses, deformations, etc.). In a two-dimensional element there are two degrees of freedom in every node: $u$ (displacement in the $x$-direction) and $v$ (displacement in the $y$-direction). To know the values of these degrees of freedom for points different from the nodes Abaqus makes use of interpolations. The linear interpolation elements are characterized for being less accurate than those quadratic and because their boundaries always keep straight during the deformation. In practice, elements with nodes just in their vertices are linear. On the other hand, apart from the mentioned advantage, quadratic elements have the capability of bending, making them more versatile to adapt to the bent edges of the part during the deformation. Besides the nodes in the vertex they have other nodes positioned in the midpoint of their sides. Triangular elements (as well as tetrahedral) can use a modified second-order interpolation. As a consequence to what has been explained, a linear triangle has three nodes whereas a quadratic triangle has six nodes (those in the three vertices plus the others in the midpoints of the three sides). Linear quadrilateral have four nodes and those quadratic, eight (or nine).

Regarding the integration, there are two types: reduced and full integration. The Gauss integration permits to integrate the value of a variable throughout the whole volume of an element. Whereas the full integration provides the exact value of the integral, the reduced shows less accuracy. A high precision requires more integration points, thus, in two-dimensions, the reduced integration quadrilateral element has only one and the fully integrated, four.

Once both characteristics have been explained, it is very important to understand the advantages and disadvantages of the elements as a function of their interpolation order and integration:

- Fully integrated and quadratic → Their great advantage lies in their suitability to mesh regions with great stress gradients. A drawback is that they can suffer from a
phenomenon called volumetric locking responsible for its increasing stiffness against
deformations. Volumetric locking usually appears when there are complex stress
combinations or in materials with incompressible behavior, very common in metals
undergoing plastic deformations, as in a forging process. Hence, its use is discarded at
the moment (although the hybrid formulation may correct this defect). In two
dimensions and plane strain the available elements are CPE8 (quadrilateral) and CPE6
(triangular).

- Fully integrated and linear → In this occasion, the issue is called shear locking, which
  arises when the element is subjected to bending. The drawback is the increasing
  stiffness of the element due to the inability of its sides to bend, causing a part of the
  energy deformation to be spent in shear strain, thus, reducing the bending. In
  conclusion, for a certain deformation these elements increase the real value of the stress.
  Solving the matter is not easy. In fact, the refinement of the mesh poorly improves the
  convergence of the values under study. Due to the absence of bending in the forging
  problem its use has to be considered → CPE4 (quadrilateral) and CPE3 (triangular).

- Reduced and quadratic quadrilaterals → When they suffer from hourglassing, a
  phenomenon to be explained in the next type of elements, they do not transmit it to the
  rest of elements in the part. They are the best option for almost any kind of problem,
  excluding those with great deformations and contact problems. According to this fact
  they turn out to be inadequate to the forging problem, as there are great deformations
  and contact in the process → CPE8R (quadrilateral). Abaqus does not use reduced
  integration in triangular elements.

- Reduced and linear quadrilaterals → The existence of only one integration point along
  with the inability of the sides to bend cause the element to have almost no resistance to
deformation; actually, its stiffness is incredibly small, so it deforms with ease. In other
  words, the stresses will be lower than the real ones. This phenomenon is called
  hourglassing and is characterized by propagating to the adjacent elements. Hourglassing
  happens for certain deformations not causing normal nor shear stresses in the element,
  as it can be derived from the fact that the lines across the single integration point will
  not change their length (absence of normal stresses) and their right angle (shear stresses)
either. Figure 20 shows an element undergoing pure bending and subsequent
  hourglassing. This type of deformation is a zero-energy mode as the deformation was
caused without requiring any energy. Unlike the shear locking, the refinement of the mesh can improve the convergence when the hourglassing arises → CPE4R (quadrilateral).

Figure 20: Hourglassing. The reduced and linear element is subjected to pure bending.

After this examination, the candidates to be the element type in the mesh are those fully integrated and linear (CPE4 and CPE3) and those reduced and linear (CPE4R). However, the Abaqus/CAE User’s Manual makes a final recommendation with respect to the selection. In the transmission of the forces responsible for the deformation of the quarter of the workpiece under the upper die, the contact between the surfaces of both elements takes place. In this contact situation, the Abaqus/CAE User's Manual states that the reduced and linear elements are the most suitable for the mesh. In two dimensions the only option is to use CPE4R elements. These elements are quadrilaterals in which the distortion they can undergo in the regions next to the contact surface, for high values of α, can make them undesirable. Anyway, it is the best option at the moment, so they will be used in the mesh. The prevention of hourglassing will be ensured (although there is no bending) by toggling on the hourglass option ‘Enhanced’ in the element features tab in Abaqus.

A meshing technique is the way Abaqus arranges the elements of the mesh following a determined pattern. Among the variety of meshing techniques provided by Abaqus (Free, Structured, Sweep…) the Structured Mesh was the one chosen because it fits in this problem and because the mesh is more homogeneous than in the other cases.

The same number of elements along the right and left side of the two-dimensional workpiece will be determined. The same procedure will be applied to the upper and bottom sides. Both numbers will try to not lead to great aspect ratios in the quadrangular elements used in the mesh. The aspect ratio relates the longest and the shortest length of a mesh element, being its ideal value equal to one (the squarer the element is the better the mesh quality will be). As an example, in Figure 21 a meshed workpiece (4x10) is represented for α > 0°.
It has always to be taken into account that the number of elements depends on the restriction in the number of nodes, one thousand, as it deals with a limited student version. Considering this fact along with the mentioned distribution of the mesh elements, the densest mesh was looked for without reaching 1000 nodes (Abaqus sends an error message). Being $m$ the number of elements along the left and right sides and $n$ their number along the upper and bottom sides, these were the equations used to find out the densest distribution:

\[
m \cdot n = \text{number of elements} \tag{15}
\]

\[
(m+1) \cdot (n+1) = 999 \text{ (number of nodes)} \tag{16}
\]

The goal is to maximize the number of elements to obtain the most refined mesh in which, presumably, the most accurate and definitive values of $R_x$ and $R_y$ will be obtained and, at the same time, to set the upper limit in the mesh density for the convergence study. To do so, the first derivative in the first equation will be equalized to zero (to find the maximum) after, previously, having written $n$ as a function of $m$ from the second equation. It is to be noted that $m$ and $n$ have been replaced with $x$ and $y$ in the following equations:

\[
(x + 1)(y + 1) = 999 \rightarrow y + 1 = \frac{999}{x + 1} \rightarrow y = \frac{998 - x}{x + 1}
\]

\[
xy = z \rightarrow x\left(\frac{998 - x}{x + 1}\right) = z; z' = 1 \cdot \left(\frac{998 - x}{x + 1}\right) + x\left(-1\right)(x + 1) - (998 - x) \frac{(x + 1) - (998 - x)}{(x + 1)^2}
\]
Then, the maximum number of elements will be \( 30.61^2 = 936.78 \). But as it must be an integer number, its value is rounded off to 936. At this point, the next step is to find the distribution for this density complying with the condition of a minimum difference between \( m \) and \( n \). According to the factorization of 936, the most even distributions are 24x39 and 26x36. For 24x39 the total number of nodes would be \((24+1)(39+1) = 1000\), which exceeds the program node limit. In the 26x36 distribution there are 999 nodes, matching the maximum allowed in the Abaqus Student Edition 6.12-2. Therefore, the 26x36 mesh is the densest one and, at the same time, the one that permits to use all the nodes. Table 3 shows the meshes used in the convergence study. The distributions in brackets have been used when eventually convergence issues have appeared. A convergence study is correctly performed against element length, not number of elements, which is almost impossible with the student version of Abaqus, due to the limitation of number of nodes. Therefore, this convergence study is only an indication of the procedure, not achieving any reliable results.

\[
z' = \frac{998 - x}{x + 1} - \frac{999x}{(x + 1)^2} = 0 = -x^2 - 2x + 998 = 0 \rightarrow x = y = 30.61
\]

<table>
<thead>
<tr>
<th>mxn</th>
<th>number of nodes</th>
<th>number of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>4x10</td>
<td>55</td>
<td>80</td>
</tr>
<tr>
<td>8x10</td>
<td>99</td>
<td>160</td>
</tr>
<tr>
<td>8x20</td>
<td>189</td>
<td>320</td>
</tr>
<tr>
<td>16x20</td>
<td>357</td>
<td>640</td>
</tr>
<tr>
<td>16x40</td>
<td>697</td>
<td>1280</td>
</tr>
<tr>
<td>24x38</td>
<td>975</td>
<td>1824</td>
</tr>
<tr>
<td>(23x40)</td>
<td>984</td>
<td>1840</td>
</tr>
<tr>
<td>(25x37)</td>
<td>988</td>
<td>1850</td>
</tr>
<tr>
<td>26x36</td>
<td>999</td>
<td>1872</td>
</tr>
</tbody>
</table>

Table 3. Meshes for the convergence study

In any case, there is a possibility to modify them if the mesh does not respect certain geometric conditions (aspect ratio and maximum and minimum angles) because of the shape and dimensions of the workpiece under study.

5.2.4 Material
With respect to the metal behavior throughout the forging there are two potential models able to represent it: the ideal-plastic solid model (Figure 22.a) and the strain-hardened solid model (Figure 22.b) (Groover, 2004).
In Eq. (2), $P/2k$ only depends on the geometric features of the workpiece and the forging conditions (temperature and lubrication that influence the friction coefficient). So the pressure $P$ will result from the use of $k$ (shear stress limit). Therefore, as the goal is to check the suitability of the calculation in $P/2k$, the simplest model is going to be chosen: the ideal-plastic model for which, once the yield stress is reached, the stress keeps constant in the plastic region, irrespective of the deformation.

In Abaqus, the characteristic elastic and plastic parameters of the material must be specified. For the different simulations the same material was used. The plastic region is determined with points representing the stress and the plastic deformation, which in addition to the elastic deformation provides the total deformation. Abaqus connects these points with straight lines and in the last one it draws a horizontal line representing a constant stress. So in the case of an ideal-plastic model it is sufficient to use just one point with a stress equal to the yield stress, $\sigma_Y$, and a plastic deformation equal to zero in the plastic region:

- Elastic Region $\rightarrow E = 2.1 \times 10^{11} \text{ MPa}, \nu = 0.3$
- Plastic Region $\rightarrow \sigma_Y = 380 \text{ MPa}; \text{ Plastic strain} = 0 \text{ at yielding}$

5.2.5 Contact

In Abaqus, the specification of the contact is capital. In fact, the program does not automatically detect its existence although the two surfaces coincide, but it will have to be the user who explicitly notifies it to the program. The first step to make Abaqus understand there is a contact between the upper surface of the workpiece and that of the upper die is the definition of the interaction. Due to the density of the theories about contact and the limited time to go deeper, the selections about contact made in a similar problem in the Getting Started With Abaqus: Interactive Edition were adopted for the forging problem. And thus, for the tangential behavior of the materials in the contact...
interface a penalty friction formulation was chosen, when considering a kinetic friction coefficient. Otherwise, it will be the frictionless option to be picked. For the normal behavior it was chosen the Hard Contact with a Penalty constraint enforcement method. Nonetheless, only the friction model (interaction) has been defined so far and not the contact detection.

To make the contact possible, the master surface (upper die) and the slave surface (workpiece) were selected to use the ‘Surface-to-Surface’ contact method with a finite sliding formulation. The finite sliding formulation was chosen in contrast to the small sliding formulation because the sliding between the surfaces was expected to be considerable with respect to the characteristic size of the elements in the contact surface.

5.2.6 Translation of the process to Abaqus
Variable conditions during the forging process simulated in Abaqus are reflected in Steps. In the analysis (the procedure ranging from the definition of the upper die-workpiece assembly to the display of the results) two analytical steps have been specified along with the default step 'Initial', in which the boundary conditions are determined. The summary of the different stages for the whole process of forging in Abaqus is shown in the next steps:

- Initializing Contact → This step is created to ensure that the surfaces remain stuck before the upper die moves down to forge the metal. Thus, there will be a continuous contact by means of the application of a very small pressure (1000 Pa) that will not affect the results. Otherwise, there could be a phenomenon called chattering in which, alternately, the two surfaces separate and get back in contact causing convergence errors (not to confuse the convergence in the program with the convergence in the results for the comparison)

- Forging → The upper die moves a prescribed distance implemented by the user. Unlike other typical Abaqus problems, there is not a force applied on the assembly. The method consists in obtaining the force exerted by the forge on the workpiece as a result of its descent and the reactions generated between the parts (the measures of \( R_x \) and \( R_y \) are made in the reference point of the solid rigid)

In Abaqus, it is of extreme importance to choose the kind of procedure of each step: General or Linear Perturbation. General steps include linear and non-linear problems.
Provided that the problem fulfills one of the following conditions, the Step will be considered General:

- Non-linear boundary conditions → Whenever contact conditions exist in the problem, the step has to be defined as General. In the current forging analysis, the contact conditions are defined since the step named ‘Initial’. Then, all the subsequent analytical steps (Initializing Contact and Forging) have to use General (Static) procedures. Even if the matter about the procedure type has been settled, it is necessary to explain the other two sources of non-linearity.

- Non-linear materials → A linear material is that one, as its name suggests, having a linear behavior. This happens only to those materials with an elastic behavior. For a material being plastically deformed there is no linearity between the stress and the deformation. In short, the metal to forge, is non-linear. So, it endorses the preceding selection made for the procedure type in the steps.

- Non-linear geometry → Although the non-linearity has already been confirmed, it is necessary to know whether there would be linearity or not in the geometry. It is said that if the geometry changes in such a way that the angles of the forces are considerably modified or if the deformations are large, then the problem will be classified as non-linear and the option Nlgeom will be turned ‘On’, as it should be in the present study due to the great deformations. But, in the beginning, as it will be seen in the following results, the Nlgeom was not turned ‘On’ because it was thought that the non-linearity in the geometry had only to do with change in the angles of the forces, something that does not occur in the forging.

5.3 Evaluation of the results in Abaqus and solutions

The following list contains the drawbacks encountered in the results with CPE4R elements, the designed distribution and the contact conditions selected:

- As it was previously said, before accepting the results of $R_x$ and $R_y$ shown by Abaqus it is necessary to check the convergence to confirm their accuracy. Both, $R_x$ and $R_y$ are projected on the perpendicular direction to the contact surface to have the force to be divided by the area resulting in the forging pressure. But the results did not converge, not in a clear way along the mesh refinements. As an illustrative example, these were
the results for a workpiece with $F_t = 1$, $\alpha = 10^\circ$, $\mu = 0.1$, $def = 0.5$ m. 23x40 and 25x37 meshes were used to highlight the convergence issues.

<table>
<thead>
<tr>
<th>mnx</th>
<th>number of nodes</th>
<th>number of elements</th>
<th>Rx (MN)</th>
<th>Ry (MN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4x10</td>
<td>55</td>
<td>40</td>
<td>-21.93</td>
<td>-105.07</td>
</tr>
<tr>
<td>8x10</td>
<td>99</td>
<td>80</td>
<td>-12.01</td>
<td>-103.22</td>
</tr>
<tr>
<td>8x20</td>
<td>189</td>
<td>160</td>
<td>-11.07</td>
<td>-103.15</td>
</tr>
<tr>
<td>16x20</td>
<td>357</td>
<td>320</td>
<td>-15.41</td>
<td>-102.42</td>
</tr>
<tr>
<td>16x40</td>
<td>697</td>
<td>640</td>
<td>-14.62</td>
<td>-102.43</td>
</tr>
<tr>
<td>24x38</td>
<td>975</td>
<td>912</td>
<td>-17.97</td>
<td>-102.26</td>
</tr>
<tr>
<td>23x40</td>
<td>984</td>
<td>920</td>
<td>-17.13</td>
<td>-102.27</td>
</tr>
<tr>
<td>25x37</td>
<td>988</td>
<td>925</td>
<td>-18.04</td>
<td>-102.24</td>
</tr>
<tr>
<td>26x36</td>
<td>999</td>
<td>936</td>
<td>-18.42</td>
<td>-102.23</td>
</tr>
</tbody>
</table>

Table 4. Workpiece analysis and values of $R_x$ and $R_y$ for different meshing.

As it was expected according to the sign of the angle, $R_x$ and $R_y$ are both negative. $R_y$ appears to converge pretty well, which is even clearer in the chart of Figure 23.

![Figure 23: Chart displaying the convergence in $R_y$ ($R_y$ vs number of elements).](image)

But this does not apply to $R_x$, which does not properly converge, as it can be seen in the chart Figure 24. In fact the same tendency was shown by other positive angles: good convergence for $R_y$, bad for $R_x$. 
Here is another example for the ‘same’ workpiece with a final shape factor $F_t = 5$. Again, the first chart is for $R_y$ (Figure 25) and the second one is for $R_x$ (Figure 26):

It should be clarified that the charts above do not reflect a proper convergence study, as it must be made plotting the value of the variables ($R_x$ and $R_y$) against the element length. But since the elements of the same mesh have different lengths (when $\alpha \neq 0^\circ$)
and the variables are global, it is believed not appropriate to choose the size of a specific element – e.g. that of the right upper corner - along the different meshes.

- Despite subjecting different workpieces to considerable deformations, the total plastification is never achieved (except for 0°). The screenshot in Figure 27 shows how the red color, corresponding to the yield stress, does not occupy the full workpiece. The other regions are elastically deformed.

![Figure 27: Workpiece after forging not showing a total plastification.](image)

Furthermore, when a considerable number of simulations have been performed, it is discovered, by chance, that Abaqus is using an automatic ‘scale factor deformation’ less than 1 in the deformed workpieces (in the screenshot above it is applied) that smoothed the actual deformations. When the unity value is applied in the factor it highlights the existence of other problems for the CPE4R elements.

- Huge distortions in elements when $\alpha \neq 0^\circ$ or $\mu \neq 0$.

- The workpiece exceeds the upper die surface in determined mesh distributions, which in practice would mean the invasion of the former in the latter (like in Figure 28). A Hard Contact normal behavior with a Penalty constraint enforcement method was applied).
Figure 28: The workpiece violating the contact at the upper die.

- The bottom right corner barely moves along the x-axis for negative values. Its displacement is negligible when it should be significant.

Figure 29: The undeformed workpiece (grey) vs after deformation (green). The displacement of the bottom right corner is too small.

The phenomenon encountered appears to be shear or volumetric locking, as seen in Figs. 28 and 29. In both figures, it looks like shear locking, but this should not be the case, since the elements are chosen not to be prone to that behavior. A possibility exists that volumetric locking is occurring. Disencouraged by the results, other alternatives (contact, mesh distribution, element type, etc.), to solve the actual problems, are looked for. Contradicting the Abaqus/CAE User’s Manual recommendations that suggest the use of reduced integration and linear elements, triangular elements are chosen (for which only full integration is considered in Abaqus) just to try other options and
because they adapt very well to the contour of the workpiece when \( \alpha \neq 0^\circ \). Linear triangles are preferred to quadratic ones to avoid the typical volumetric locking of incompressible metals. That is, the selected elements are CPE3. One advantage in using them is that the number of elements is doubled up with respect to the quadrilateral elements (the quadrilaterals are divided into two triangles by tracing their diagonal). The fact that the aspect ratio is now a little bit worse than before, the diagonal is longer than the longest side of the quadrilateral element, is negligible in comparison to the improvement in the mesh refinement. The Abaqus manuals advise against using this type of elements provided that they are not sufficiently refined, but a mesh with 1872 elements (in the case of the 26x36 distribution) seems sufficiently dense. The improvements, waiting for the verification of their validity, are noticeable in Figure 30:

- The upper surface of the workpiece does not exceed that of the upper die anymore.

- The workpiece is totally plastified, which allows comparing the Abaqus results to those of Eq. (2).

![Figure 30: The workpiece meshed with CPE3 elements. Total plastification achieved and violation of the contact of the upper die avoided.](image)

- The bottom right corner moves significantly, as it does the upper right corner too (Figure 31).
In the verification of the triangular mesh, the frictionless contact condition is determined as the final reference to assess the suitability of the mesh and the correct selection of the contact parameters with which to perform the further analysis for all the workpieces irrespective of their angles ($\alpha$) and friction coefficient ($\mu$). So, besides the total plastification in the deformed workpiece, the criterion to approve the meshing and the contact conditions, as well as the friction model, is to check the nearly incompressible behavior of the metal during the deformation. Having fulfilled these requirements in a frictionless workpiece, it is possible to conclude that the contact options and the meshing chosen are correct and that, in all probability they will lead to satisfactory analysis for any other configuration, even implying nonzero friction coefficients. This stage is, thus, very important for confirming the starting parameters in the analyses which will then be transferred to other analyses, even to those with a nonzero friction coefficient.

In the study of a workpiece with $\alpha = -15^\circ$, $F = 1$, an upper die descent of 0.5 m ($def = 0.5$ m) and no friction ($\mu = 0$) there are two configurations of the upper right and the bottom right corners for the different meshes of the workpiece. In Fig. 29, in the screenshot on the left, the bottom right corner displacement is bigger than that of the upper right corner, while on the right screenshot the opposite occurs.
Considering that the right side of the workpiece, after the deformation, is almost straight, two equations to calculate the area variation were derived for the two configurations shown in Figure 32. In both cases, the area decreases a 19%, which contradicts the incompressibility of the metal. The conclusion is that CPE3 elements do not resolve the problem satisfactorily. Then, a new approach has to be made, coming back to the recommended CPE4R elements.

In fact, reconsidering the option of including the non-linearity in the geometry, the problem of the workpiece invading the upper die disappears, as it can be seen in Figures 33 and 34.
Figure 34: Workpiece exceeding the upper die with Nlgeom = Off.

According to Figure 33, despite the improvement achieved, there will not be a total contact in negative angles and, thus, the workpiece will not be totally plastified.

The deformations, using a regular mesh, are always huge in the forging simulation. In the late stages of this Final Year Project, it was discovered an Abaqus feature likely to solve all the problems experienced so far: ALE (Arbitrary Lagrangian-Eulerian) adaptive meshing. Adaptive meshing is thought for those processes where there are huge deformations responsible for great distortions in the mesh elements. With this technique the mesh will not move with the greatly deformed material, thus it will keep a high quality during the forging simulation.
6. Conclusions and discussion

Despite the goal to check the validity of the equations for the forging pressure, by comparing their results to those provided by the finite element method, it has been finally impossible to achieve the goal satisfactorily due to the difficulties arose because of the nature of the process, the limitation in the Abaqus Student Version used and the lack of time to go deeper in the study of alternative and more sophisticated solutions. One important conclusion is believed to be that validating a physical phenomenon as forging with software, is a tedious task and easily exceeds the frame set by a Final Year Project.

In a forging process huge deformations take place, causing such distortions in the elements that the regular Abaqus analysis fails to offer accurate results. In similar situations, an appropriate solution to improve the results is to refine the mesh. However, taking into account that it was used a student version of Abaqus limited to one thousand nodes, the possibilities in using that technique were restricted. Neither the local refinement (partitions) nor the rounding (fillet) of the stress concentration corners of the workpiece improved the results. These and other alternatives (e.g changing the element type) have been implemented without success although, at the same time, they helped to delimit options and to orient further analysis to a new technique discovered lately while performing the present Final Year Project: ALE adaptive meshing. This technique seems suitable for the forging analysis because, as the theory states, the mesh keeps high performance (along with a good refinement), being its movement independent of that of the material. Thus it is believed to be the appropriate for processes with great deformations like the forging. An additional difficulty was the contact between the workpiece and the upper die, since it requires wide and deep knowledge in the field to correctly choose the model that governs its behavior (normal and tangential) not to mention its detection.

A procedure to collect the values of the expression $P/2k$, as a function of the equations variables along with the limitations in the shape factors to use according to the angle, has been determined. Also a method by which to determine the initial measures of the workpiece, fixing the descent of the upper die and the shape factor at the end of the process, to ensure the plastification has been drawn. Finally, the basis for further
analysis has been set: CPE4R elements, Nlgeom = ON, kind of contact, ALE adaptive meshing, etc.

Not less relevant is the divulgation of the equations, and their corresponding complex theory, in such a simplified way (accompanied by examples) to make them attractive to researchers interested in finding improvements in the forging field.

Last, but not least, the recommendation for further validation of Eq. (2) is to perform physical experiments to study relevant behavior of the material and the forging parameters.
7. Future work

The future lines, with respect to the validation and subsequent deepening into the theory for further improvement of the equations, will have to deal, first, with the ALE adaptive mesh solution for the Abaqus analyses. This would be the very first step in the short-term procedures. Once the analyses will have been satisfactorily carried out, the validity ranges of the equations will have to be determined. A first approach to the verification of ‘the equations vs FEM’ will be by charts, where the angle and the friction coefficient would have been fixed, confronting $P/2k$ (y-axis) against the shape factor $F$ (x-axis) for both the equations and the FEM analysis. Workpieces with $F_i < 1$ will have to be discarded, since when $F$ tends to 0 in the equations $P/2k$ tends to infinite. The reading of the chart will be from the left to the right, according to the evolution of the workpiece in the forging (increase of F). An increase of $P/2k$ is expected as long as the forging progresses, so the values provided by the equations will have to satisfy this simple condition in all the comparisons. The power throughout a real forging constantly grows from the beginning of the process. It would be interesting to find, at least, a big validity range for $F$, flanked by $F < 1$ and $F >> 1$. On the contrary, there would be discontinuities generating uncertainty regions in the application of a pressure to the machine, although it could be interpolated from the extreme values of the two valid ranges. It will be compulsory to establish a method to discard or accept the results based on technical features (power administration) of the most common machinery used by the target forging companies.

The next challenge for the equations would be a workpiece with multiple modules, since two important issues would be under study: the correct modeling of the material fluency from one module to another and the global equation of $P/2k$. The analyses would be performed for the simplest situation, two modules, which will generate countless combinations. In case there is any error, it would be the moment to check the theoretical basis of the equations.

For workpieces with contours able to be divided in multiple modules, it is essential the design of algorithms with which to have the best modular division (that one leading to the lowest value of $P/2k$). This will be an important procedure in long-term actions because, along with the two-module workpieces, the lowest the value the most accurate the comparison to the FEM analysis will be.
Bibliography and references


*Getting Started With Abaqus: Interactive Edition. Version 6.8*


http://www.instrumentacionycontrol.net/cursos-libres/automatizacion/curso-completo-de-plcs/item/105-capitulo-33-dispositivo-de-programaci%C3%B3n-o-terminal-en-un-plc.html.


Martín, F. (2009). Desarrollo, integración y optimización en el estudio de procesos de forja mediante el teorema del límite superior a través del modelo de bloques rígidos triangulares (PhD). Málaga, Spain: Escuela Politécnica Superior de la Universidad de Málaga.


