Validation Of A Purely Elastic Model And A Finite Element Model For A Screw Press

Jean-François Mull[1 a)], Camille Durand[1 b)], Cyrille Baudouin[1], Régis Bigot[1] and Marc Borsenberger[2]

[1] Université de Lorraine, Arts et Métiers ParisTech, LCFC, F-57000 Metz, France
2 Manoir Industries Bouzonville, Rue de Guerstling, F-57320 Bouzonville, France

Corresponding author:
[a)] jeanfrancois.mull@ensam.eu
[b)] camille.durand@ensam.eu

Abstract. The increasing use of superalloys in the industry brings new issues to forging manufacturers concerning Finite Element analysis of the forming process. Once we consider high performing materials, significant differences can occur between numerical and experimental results. The lack of knowledge about materials and thermal exchanges explains a part of these differences but it has been shown that the machine behavior has a significant impact on the forging process. Press modelling is a way to quantify the energy dissipated in the machine structure and improve simulations. This study is a first step in the screw press modelling. Starting from an approximation of the press geometries, a purely static elastic (PSE) model is developed and a numerical simulation considering only uprights is performed. Uprights stiffness predictions from the PSE model and the FE simulation are compared and the PSE model is validated. Then, the press is approximated by a portal frame and FE simulation is realized to determined uprights stiffness by considering links effects and bending behaviors in the structure. Comparison with experimental results validates the approximation.

INTRODUCTION

Forging processes are often developed with the help of numerical simulation to determine the best sequence and process parameters. However, for high performing materials, significant differences are observed due to the elastic collision between tools and billet caused by the high resistivity of the material [1]. Thus, an important part of the stroke energy is returned to the machine and must be properly considered to improve simulation's prediction.

During a forging operation, a part of the stroke energy is dissipated as elastic energy in the machine itself. A first approach is to quantify this loss by modelling the press as an equivalent spring [2]. It is possible to consider an equivalent spring in the three directions, axially and in rotation. With the stiffness associated with these springs, the loss of energy in the machine deformations can be calculated. Another way to consider the press behavior is to introduce the flexibility matrix, whose coefficients express the deformation of the axis *i* in rotation or axially due to a moment or a load applied on the axis *j*. This matrix can be completed with a clearance's vector to consider the non-linearity of the force due to the guidance clearances. This model is commonly used and was applied for the case of hydraulic, eccentric and screw presses [3-5]. Press stiffness identification methods are an important subject of research because of the difficulties to stress machines in the adapted way. Indeed, determining stiffness experimentally, demands to develop complex tools able to generate the needed deformation to be able then to calculate stiffness. A loading of a machine in a specific direction can be realized through two mains methods: either we use the machine in its normal functioning mode and a stroke is exploited as load generator [6,7] or external device like hydraulic cylinder placed on the press table can be used to apply load directly to the ram [8]. Otherwise some studies propose to perform FE simulation to avoid experimental errors and simplify mechanical loading [9]. Then, experimentally or numerically, deformations obtained allows to calculate press stiffness. Several authors proposed to predict the equivalent press stiffness by modelling the press as a succession of springs [5,10]. Each spring represents an element of the press with its proper stiffness. The equivalent press stiffness is expressed according to the press elements stiffness and the type of elements connection defined as series or parallel. So far, this method considers a limited number of press elements and do not provide a specific way to obtain press parts stiffness. This paper presents a purely static elastic (PSE) model for a screw press based on the elastic behavior of materials and an approximation of the press geometries. This model allows to predict the press uprights stiffness. Moreover, the CAD of the uprights is developed, and FE simulation are performed to obtain stiffness. Then, the CAD is completed with the crosshead to approximate the press by a portal frame. FE simulation is realized and the stiffness value obtained is compared with experimental investigations with the aim to validate the approximation of the press by a portal frame.

MATERIAL

The screw press of our study is the LASCO SPR 400 from the VULCAIN platform in Metz, in France (Fig. 1 a)). The structure of the machine is a portal frame made of four uprights constituted of pre-stressed steel tie rods embedded in a cast iron case. A bedplate bears the four uprights which support the crosshead. The press ram is linked to the screw with a nut fixed to the ram. The screw being linked to the frame by a pivot linkage, this assembly allows a translation motion for the ram. The rotation of the screw is ensured by the flywheel on the top of the press, alimented by a direct electric drive motor. The motor can be reversed to allow downstroke and upstroke. Concerning materials, the screw and the ram are made with steel whereas the nut is made with bronze. The press integrates strain gauges fixed on the press crosshead to measure the evolution of the load during strokes. Furthermore, the position of the ram in comparison with its bottom dead point is recorded with a magnetic coding system.

VALIDATION OF THE PSE MODEL

Purely static elastic model

Press geometries are approximated by using standard forms defined in TABLE 1. The schema of the approximation of the machine is shown in Fig. 1 b).

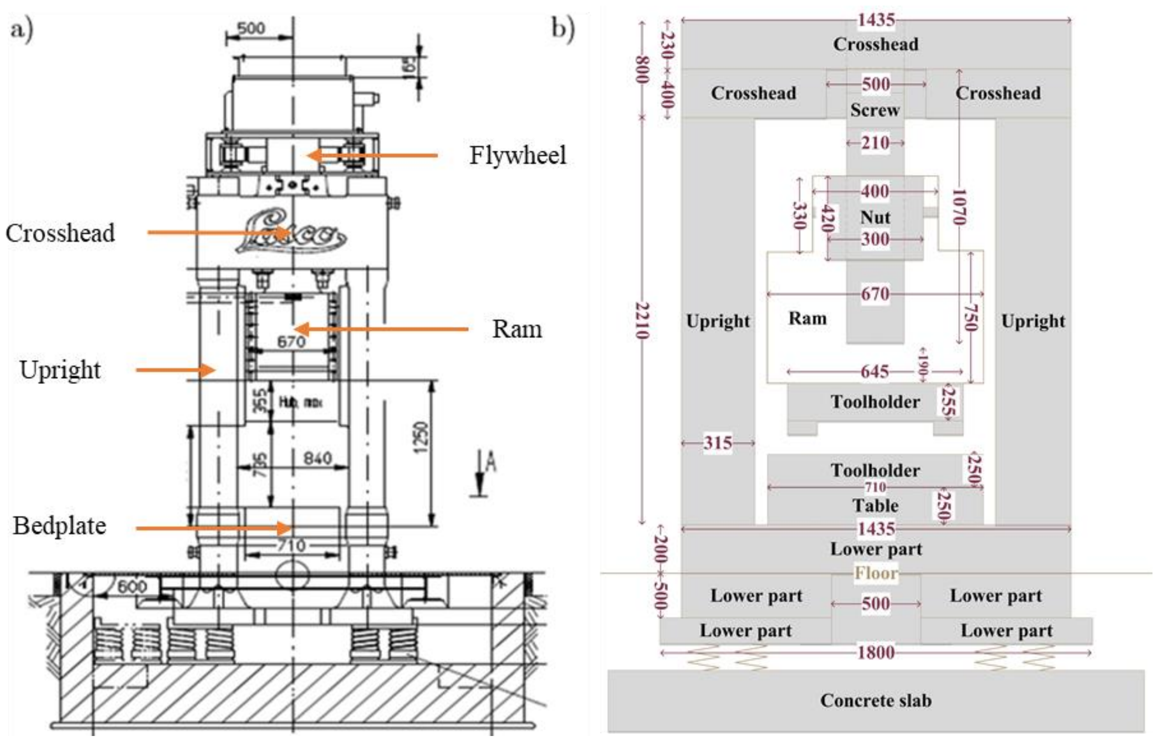


FIGURE 1. a) Face view of the press LASCO SPR400 b) The geometrical approximation of the press elements

The standard forms chosen to discretize the press are defined with a surface S and a length l_0. Considering one press part, according to the Hooke’s law:

$$\sigma = E.\varepsilon \tag{1}$$

With σ the stress applied on the surface S, ε the equivalent strain and E the Young’s modulus of the material. By substituting the stress and the equivalent strain by their definitions in the Eq. (1), an expression of the load applied on the surface S can be obtained as in the Eq. (2).

$$F = \frac{E.S}{l_0}.(l - l_0) \tag{2}$$

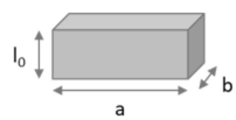
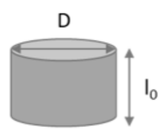
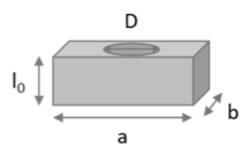
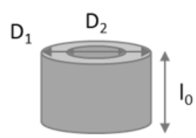
With F the load applied on the surface S and l-l_0 the true stain of the element. Given the Eq. (2) and the relation between load and deformation for a spring Eq. (3), the stiffness of the element can be identified Eq. (4) then calculated according its material and its geometry.

$$F = k.(l - l_0) \tag{3}$$

$$k = \frac{E.S}{l_0} \tag{4}$$

For each one of the forms defined in the TABLE 1, it is possible to calculate an associated stiffness thank to the Eq. (4). Thus, it is possible to determine the stiffness of each press part considered.

TABLE 1. Stiffness of standard forms

Geometries	Diagram	Stiffness
Cuboid		$K_{Cuboid} = \frac{E.a.b}{l_0}$
Cylinder		$K_{\text{Cylinder}} = \frac{\pi.E.D^2}{4.l_0}$
Drilled cuboid		$K_{Drilled\ cuboid} = \frac{E.(a.b - \frac{\pi.D^2}{4})}{l_0}$
Tube		$K_{\text{Tube}} = \frac{E.\pi.({D_2}^2 - {D_1}^2)}{4.l_0}$

By studying the connections between each element, the press can be modeled as a succession of springs. In the Fig. 2, each line represents an element whereas points show the type of connection, series or parallel. For each one of these elements is associated a proper stiffness. Crosshead and bedplate are not yet modelled.

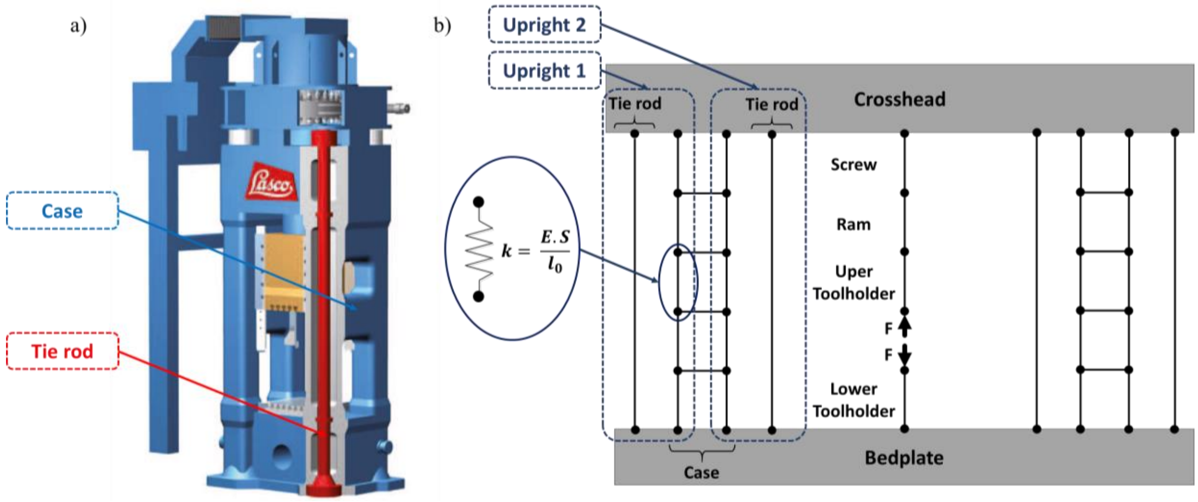


FIGURE 2. a) The structure of the screw press LASCO SRP400 b) Purely static elastic model of the press system

This study focuses on the calculation of equivalent uprights stiffness. Each upright is constituted of one pre-stressed steel tie rod and a cast iron case common between two uprights, as it is shown in the Fig. 2. According to the type of connections and the stiffness of each spring composing the uprights, the equivalent stiffness of the four uprights is calculated: $K_{PSE} = 2.39*10^{10}$ N/m.

Finite element simulation

The CAD is developed with CATIA V5 to realize FE simulations with the module Generative Part Structural Analysis. The mesh is defined as regular and a 50mm size parabolic tetrahedron element is chosen. First, the uprights are modeled considering the steel tie rods and the cast iron case (Fig. 3 a)). In the FE simulation a tensile load of 5000kN is applied on the top of the uprights, the base of the uprights is fix. The stiffness of the uprights is calculated according to the Eq. (3) with the vertical elongation of the uprights from the results of the FE simulation: $K_{UprightsFE} = 2.45*10^{10}$ N/m.

Results

TABLE 2. Stiffness obtained by PSE model and uprights FE model

PSE Model (N/m) *10^{10}	Upright FE (N/m) *10^{10}	Standard deviation
2.39	2.45	2.4%

The PSE model and the simulation shows a good agreement. Indeed, the relative deviation between k_{PSE} and $K_{UprightsFE}$ is equal to 2.4%. This small deviation can be explained by the approximations done in the geometries and materials definition of the press elements in the models. Thus, in the case of axial load, the PSE model is a good way to predict press part stiffness, even for complex structure as the uprights. It might be possible to extend this approach for other press elements.

VALIDATION OF THE PORTAL FRAME APPROXIMATION

Finite element simulation

The CAD of the uprights is completed with the crosshead to approximate the behavior of the whole press by a portal frame (Fig. 3 b)). A FE simulation is realized with the same mesh definition as the first simulation. A load of 5000kN is applied on a circular surface centered on the inferior part of the crosshead, corresponding to the screw surface, this simulating the stress caused by a forging operation. Then, the stiffness of the uprights is calculated in the same way as previously: $K_{PortalFrameFE} = 1.40*10^{10}$ N/m.

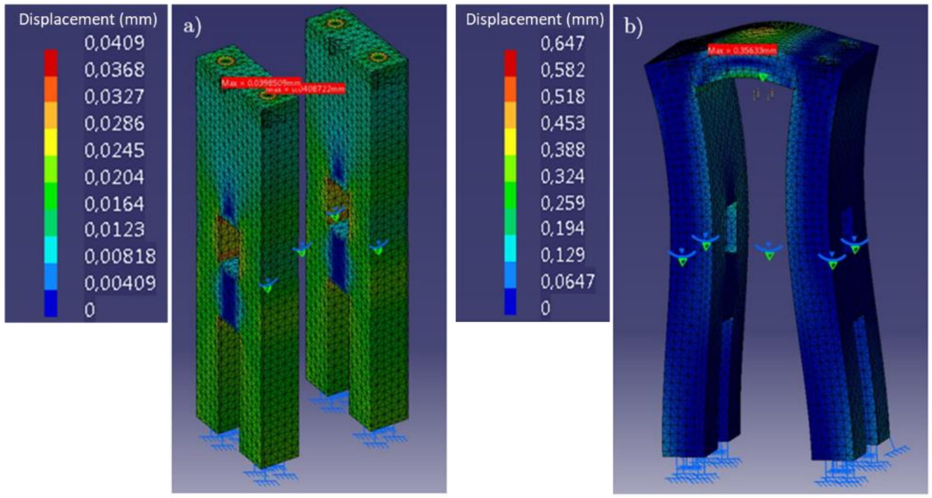


FIGURE 3. a) Result of the finite elements simulation of the press uprights b) Result of the finite elements simulation of the press approximated by a portal frame

Experiments

The determination of the uprights stiffness necessitates the measurement of the uprights elongation during a stroke. For this purpose, a 3D tracking point system following reflecting stickers placed on the upright was used (Fig. 4 a-b)).

Moreover, in order to measure a significant elongation of the upright, load applied on the frame must be maximal and the forming time must be long enough to allow the recording of enough data. Inelastic collision on C17 carbon steel billet implying a maximal force of 6414 kN was chosen. This load is closed to the maximal load allowed by the machine. In addition, the billet is designed with a high length/diameter ratio to increase forging time and record more data.

Experiment was repeated twice, load and elongation curves are overlapping for both. Thus, the stroke is considered repeatable and only the first experiment is shown for clarity reason (Fig. 4 c)).

However, data recorded by PONTOS cannot be used directly. Indeed, PONTOS measures the distance between two stickers but not the total elongation of the upright. A linear deformation of the upright is assumed to calculate the total elongation.

Knowing the load applied and the elongation, the uprights stiffness is calculated with the Eq. (3) for each measurement. Then, the mean of these values is calculated: $k_{Exp} = 1.31*10^{10}$ N/m.

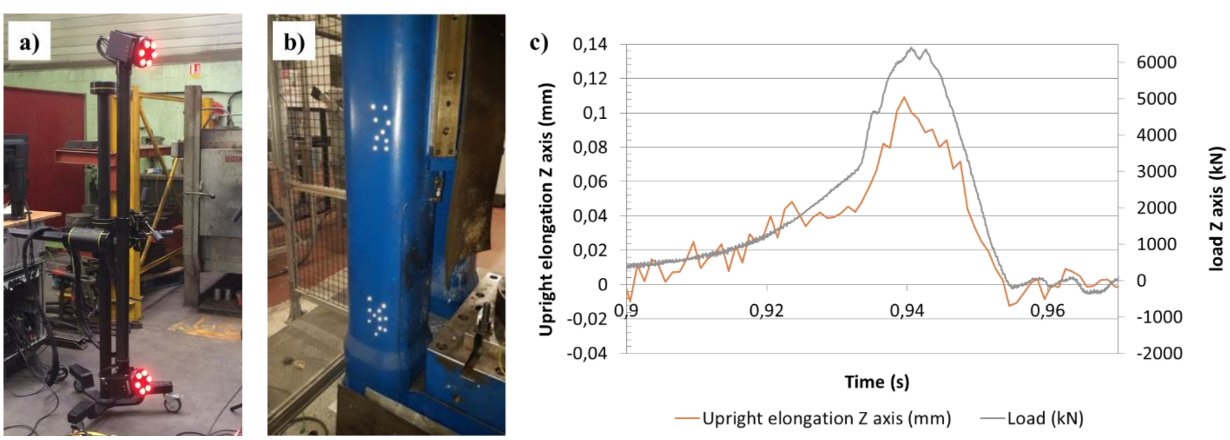


FIGURE 4. a) 3D tracking point system PONTOS b) Stickers placed on the press upright c) Upright elongation and load during a stroke on carbon steel C17

Results

TABLE 3. Stiffness obtained by portal frame FE model and experiment

Portal frame FE simulation (N/m) $*10^{10}$	**Experiment (N/m) $*10^{10}$**	**Standard deviation**
1.40	1.31	6.9%

We notice that the FE simulation of the portal frame agrees well with the experimental results, indeed the relative deviation between $K_{PortalFrameFE}$ and K_{Exp} is equal to 6.9%. Sensors accuracy and approximations on the geometries and materials definition of the press elements can explain this slight difference. Despite the simplicity of the portal frame approximation to model the screw press, the FE simulation provides a good prediction of the experimental uprights behavior. The use of portal frame approximation in FE simulation could be a way to predict vertical load and moments applied to the uprights by considering the links effects in the structure and the bending behavior of the crosshead. A coupled approach between FE simulation and analytical modelling could be a way to improve the PSE model by considering links effects and bending behavior.

CONCLUSIONS

Considering a screw press as a succession of springs, this study proposed to predict upright stiffness with a purely static elastic model based on press geometries and materials. FE simulations for the press uprights and the whole press approximated by a portal frame were performed. The results of simulations and experimental investigations allowed us to determine several uprights stiffness values. The following conclusions were drawn:

1. PSE model is sufficient to explain uprights behavior without the consideration of the crosshead. If links effects and bending behavior can be neglected, it might be possible to predict stiffness of other press elements with the PSE model;
2. An approximation of the press by a portal frame allows good predictions of the press uprights stiffness with FE simulation. It may be an appropriate solution to predict moments and load applied to the uprights;
3. A step forward to improve the prediction of the PSE model will be to consider the crosshead bending behavior and links effects in the press structure. FE simulation of the screw press by a portal frame could be a way to study these effects and complete PSE model.

REFERENCES

1. C. Durand, R. Bigot, and C. Baudouin, 17th International Conference on Metal Forming, Metal Forming 2018, 16-19 September 2018, (2018).
2. VDI 3193, (1986).
3. K. Chodnikiewicz, R. Balendra, and T. Wanheim, Journal of Materials Processing Technology 44, 293 (1994).
4. M. Arentoft, M. Eriksen, and T. Wanheim, Journal of Materials Processing Technology 105, 246 (2000).
5. H. Ou and C.G. Armstrong, Finite Elements in Analysis and Design 42, 856 (2006).
6. M. Arentoft and T. Wanheim, CIRP Annals 54, 265 (2005).
7. C. Brecher, W. Klein, and M. Tannert, Production Engineering 4, 279 (2010).
8. B.-A. Behrens, C. Brecher, M. Hork, and M. Werbs, Production Engineering 1, 31 (2007).
9. B. Lu and H. Ou, Proceedings of the Institution of Mechanical Engineers, Part B: Journal of Engineering Manufacture 226, 466 (2012).
10. M. Croin, A. Ghiotti, and S. Bruschi, in (AIP, 2007), pp. 493–498.