

1 Introduction

When parts with complex geometry and high-tensility (eg. pulleys, gears, towbars, connecting rods) are required to be made, they are produced by die forging. Thus, these parts possessing the required mechanical attributes may be mass produced with a relatively low per-unit cost.

Tools must be designed with great caution with regard to high manufacturing costs and losses of production due to possible tool breakage. In favour of this, the forging process should be examined both with traditional designing methods and the finite element method.

2 Examination of the forging process

In the present work, I completed the strength endurance test of a forging die that may be suitable for manufacturing the forged blank of a gear (Figure 1). For testing purposes, I employed the finite element method, applying dynamic marginal conditions regarding the plastic deformation of the forged

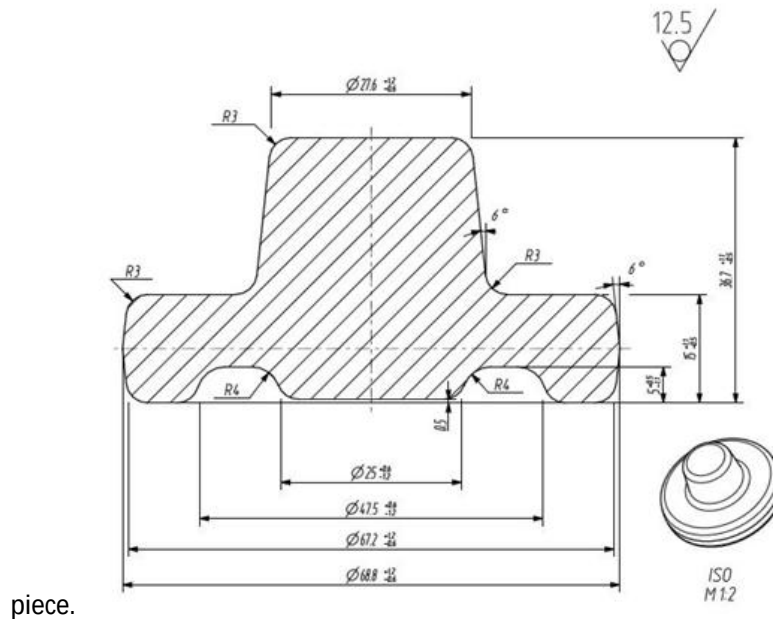


Figure 1: The forged piece

My aim was to examine the adequate filling of grooves and to define the tension in the moulded piece. The analysis was done by using AUTODYN in Explicit Dynamics module of Ansys 12.1 Workbench 3D CAE design suite. This special module of the software is capable to simulate those processes that take place within a short time and may cause considerable deformations, even breakage or deterioration may also happen. On account of high speed and shifts, the computing requirements of these analyses are huge; therefore I strived to create the simplest model. Considering the geometrical and loading symmetry of the tested structures, one may substantially reduce the computing requirements of the tasks. The simplification was based on the fact that the points of the structures that fall into the median planes, also remain on the median planes.

2.1 Geometrical model

As both the tested tool and the starting piece have two-fold symmetry, I only modelled one quarter of them in 3D with the help of the software. The dimensions of the die halves and the starting piece are in compliance with the values defined during the designing process. I set the distance to 0.05 mm between the plane of the upper die half and the starting piece to reduce 'unnecessary' computing to minimal, namely shortening the path of the upper die half until it touches the moulding material. During constructing the geometrical model (Figure 2/b), to further reduce the computing requirements, I applied the following simplifications:

- In the die halves I only placed the finishing grooves not any bore-holes
- I left the grip and the guide pins out of consideration, I considered their effects by kinematic forces.

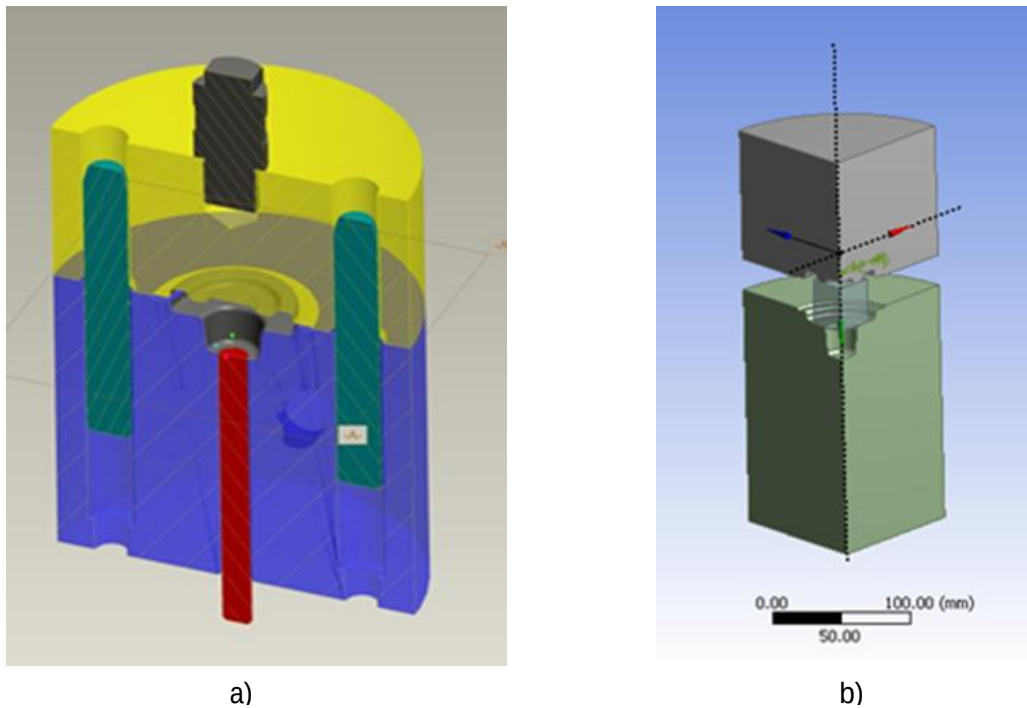


Figure 2: a) 3D model of the tool b) The geometrical model

2.2 Material model

During the course of modelling the forging process, I assumed that the starting piece retains a constant temperature (1000 °C) during the moulding. Besides, to reduce the computing requirements, I left the heat transmission between the heated piece and the tool (that is the heating of the die halves and the cooling of the forged piece) out of consideration. By altering the static attributes of the workpiece, I took its plasticity at 1000 °C into consideration by the bilinear material model. The tool halves were made of X4CrMoV5 hot-work tool steel that was hardened to ~42 HRC.

Table 1: Material models used during the examination

	<i>Starting piece</i>	<i>Tool</i>
<i>Density</i> $\left[\frac{kg}{m^3} \right]$	7850	7850
<i>Elastic modulus</i> [MPa]	40000	210000
<i>Poisson's ratio</i> [-]	0,33	0,33
<i>Yield point</i> [MPa]	40	2000
<i>Tangential modulus</i> [MPa]	20	-

During the forging process, I defined the frictional contact between the heated material and the die cavities, as well as between the connecting surfaces of the tool halves in a closed position. The values of the static and kinetic frictional coefficient are shown in Table 2:

Table 2: Values of the static and kinetic frictional coefficient

	<i>Between the heated material and the die halves</i>	<i>Between the die halves</i>
<i>Static Frictional Coefficient [-]</i>	0,05	0,1
<i>Kinetic Frictional Coefficient [-]</i>	0,03	0,08

2.3 Finite Element Model

I completed the examination on simplified models of components with a complex geometry, which are exposed to dynamic forces. With the help of the software, I sectioned the body, which models the parts, into a finite number of tetrahedrons. During the sectioning I used a finer mesh for the parts where the results may be decisive in terms of accomplishing the task and a coarser mesh where the changes may not be notable.

The items are 4-node ones so the program approximates the connection of two nodes by a linear function. The more nodes the generated mesh has, the higher the computing requirements of the task are, so when meshing the model, I was careful to use a close mesh only on those surfaces where a huge deformation takes place during the forging process or where great tensions arise (die cavities and the surfaces of the workpiece). On account of the different sizes of the items, I had to be careful to avoid coarse transitions in the mesh so I chose 3 mm as global size of items and 1.8 mm as the size of the starting piece. The refinement of the mesh is required because of the considerable deformation of the starting piece. Considering the tensions arising in the tool halves, the die cavities are the most critical places, therefore the mesh had to be refined on these surfaces. The element size was also adjusted to 1.8 mm. In the last stage, to reduce the computing requirements, I coarsened the mesh on those surfaces where critical deformations may not be expected. On the outer surfaces of the die halves, I set the element size to 15 mm. (Figure3/a)

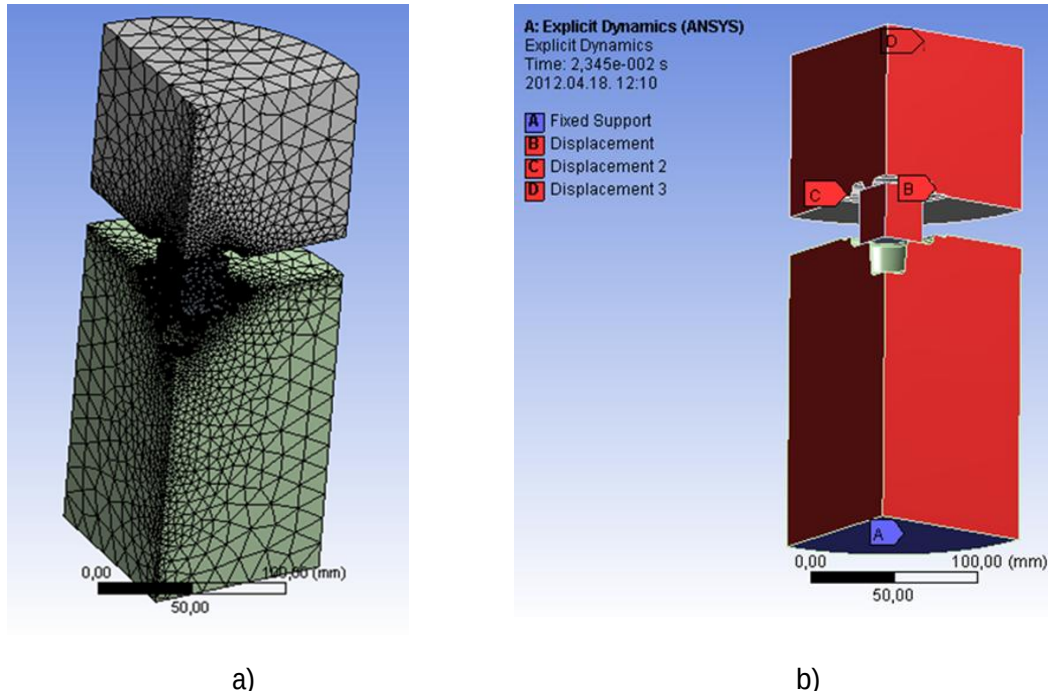


Figure 3: a) The generated mesh; b) The marginal conditions

2.4 Marginal conditions

I defined the speed of the upper die half at a constant value of 1000 mm/s, in accordance with the moulding speed based on the characteristics of the moulding machine. Subsequently, in compliance with the symmetry conditions, I set the restriction of perpendicular movements of the surfaces on the median planes. As the next step, I set adequate constraint for the fastening of the die half by restricting the surface's all degrees of freedom. In the last step, I defined the vertical shift of the upper die half. I did not restrict the shift of the surface along the direction of axes 'x' and 'z'. At the end of the process the tool would lock up completely. (Figure 3/b)

3 Results

In each body, the equivalent stress, the directional deformation and the maximum shear stress of the moulded piece were calculated. (Figure 4)

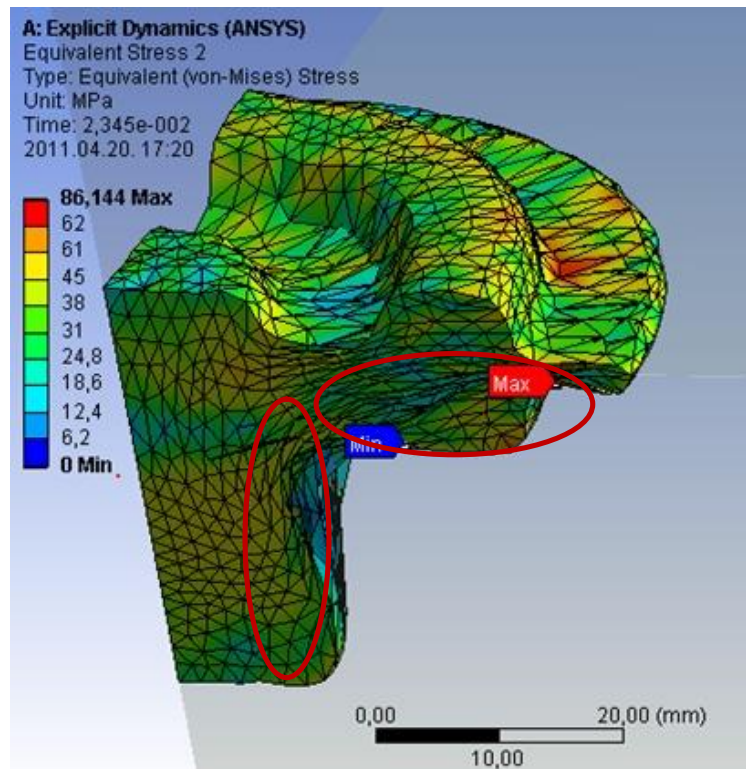
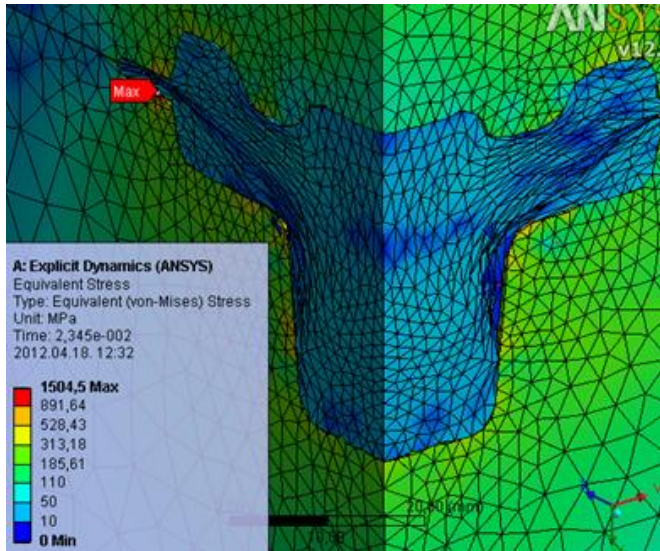


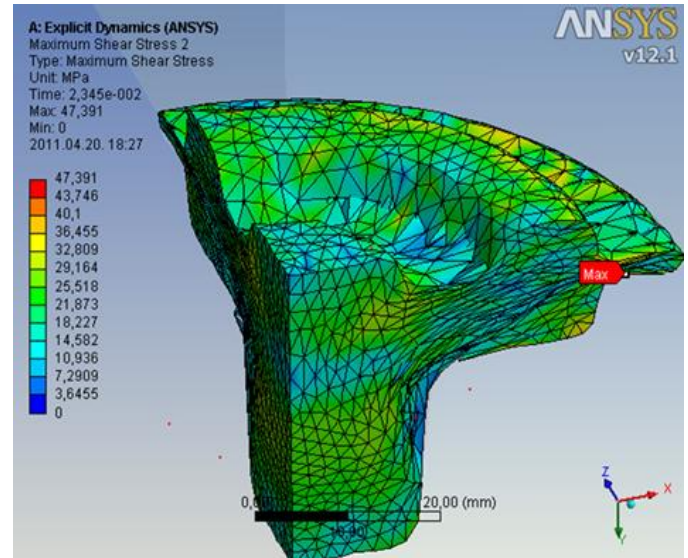
Figure 4: The maximum equivalent stress rising in the moulded piece

As a result of the upper die half's motion, the forged piece adequately fills in the available space at the end of the process. There's a sufficient amount of material in the gutter so the dimensions of the gutter are adequate and they ensure an excellent filling of grooves. The generated mesh is strained to a great extent in the sections where considerable deformation occurs, consequently, in these sections (Figure 4) grain refinement is expected. During the moulding, the material yields but no discontinuity occurs. A maximum of 86.14 MPa tension arises at the fin. (tensile strength: $R_m=100$ MPa). (Figure5/b)

As expected, the greatest tensions arise on the surfaces of the die cavities. The dimensions of the die halves are adequate; as neither die half undergoes irreversible deformation during the moulding, the maximum reduced tension is 1504.5 KPa (the yield point of the tempered die halves is : $R_{eH}=2000$ MPa). (Figure5/a)



a)



b)

Figure 5 a) The forged piece in the tool; b) The moulded piece

4 Summary

Based on the results of the simulation, the solidity of the tool is adequate; the tool is not likely to break. The blank is expected to fill the cavity adequately; respectively, according to the deformation of the mesh, one may deduce the location of grain refinement in the forged piece. For pieces with similar size and geometry, the same adjustments may be used. My further aims are to optimize the dimensions of the tool and create a completely correct modelling of the forging process by observing the aspects of thermo-mechanics.