The finite element analysis of pelvic assembly resulted from a pedestrian’s traffic accident. The dynamic analysis

I Pascal, C C Ciobotar, M Gaidur, E Rakosi, Gh Manolache and S Talif

Mechanical and Automotive Engineering Department, “Gheorghe Asachi” Technical University of Iași, Romania

E-mail: ionut.pascal@yahoo.com

Abstract. The purpose of this study is to make a series of finite element analyses of the pelvic bone assembly of a pedestrian involved in a car crash. The analyses are of a dynamic type. In this way we took into consideration a normal pelvic bone assembly and an osteoporotic pelvic bone assembly. The conditions for analyses were chosen as to reflect the reality of the situation. With the help of this study the authors want to create a data base which aims to improve the time needed for achieving an accidentology technical expertise.

1. Introduction

Accidentology is based on an unique and universal research mechanism precisely to allow the comparisons within procedures not only between the regions of a country, but also between different states, making possible taking over and distribution of those effective measures that have been checked in practice. The research and implementation activity of the results is a complex one as it requires interdisciplinary groups of specialists to certify the best solutions. Among them there are also economists who are to determine the costs and politicians who are to decide well-coordinated policy measures as well as consistent implementation of decisions into practice.

According to a study carried out by experts in road accidentology, it appears that about 75% of pedestrians involved in road traffic accidents are hit by road vehicles. The results confirm that 17% of pedestrians are hit by trucks, other 14% by buses and 5% by motorcyclists. The severity of injuries resulted from the impact depends mostly on the type of the vehicle involved. In most of the cases the pedestrian is hit by the front of the vehicle, this means by the bumper [1].

In some situations when, unfortunately, the accident results in the death of the pedestrian it is mandatory to carry out an accidentological expertise, which aims to determine the speed of the vehicle involved. In order to determine this, technical experts are using several methods (mathematical calculus using formulas from the literature, PC. CRASH, ANSYS, et cetera) [2].

2. The theoretical basis of the study

The situation described in this study is this: the car is driving at a speed of 70 [Km/h], from various reasons he does not see a pedestrian who intends to cross the street through at a zebra crossing. As a result of this incident the pedestrian suffered a series of injuries on the body [2],[6].

The main focus of this study is on every fractures or injures that may occur on the pelvic bone assembly in the situation described above. With the help of the ANSYS program we performed two dynamic finite element analyses. For both finite element analyses we used a 3D scanned image of a pelvic bone assembly [4]. The mechanical properties of the pelvic bone assembly which are shown in
From the literature [3], [5], we must mention the fact that all the bone structure is considered to be a homogeneous one, for both types of bones.

**Figure 1.** The mechanical properties of pelvic bone assembly

Figure 2 shows us the initial set-up introduced in the ANSYS program. In order to run the finite element analyses in a shorter time, we decided to consider only the bumper of the vehicle, the pelvic bone assembly and the ground. As we can see in figure 2 the ground is represented by 2 surfaces: the horizontal one represents the walking way of the pedestrian and the vertical one represents the area where the pedestrian is projected after impact [7].

We chose this hypothesis for reasons related to the power of the computer used for performing the finite element analyses.

The pelvic bone assembly was divided into a number of 93,333 elements, represented by the help of 19,259 nodes. The vehicle speed is 70 [Km/h] and the mass is 1400 [Kg].

**Figure 2.** The initial conditions for the analysis
3. The finite element analysis. Results and interpretations

Following the finite element analyses of the impact between the vehicle and the pelvic bone assembly, the images presented below are shown. The scales of equivalent stress are represented in figure 3 for normal pelvic bone assembly and figure 5 for osteoporotic pelvic bone assembly. The figures 4 and 6 show us the stress distribution into pelvic bone assembly structure. Helped by details a, b, c and d we highlighted the risk areas for each type of pelvic bone assembly.

Analyzing the scale of equivalent stress from figure 3, we notice that on the top the value is 912 [MPa]. For this value we assigned the red color. At the bottom of the scale we notice the value of 0.12 [MPa], for which we chose the blue color.

Looking at figure 4 we notice that the highest value of equivalent stress is met on the front part of pelvic which was hit by the vehicle, but the most affected area is on the side of the pelvic which took contact with the ground.

The area captured in detail A, highlights the part of the iliac bone, as being the first area of the pelvic bone assembly that is affected by the impact between vehicle and pedestrian. The value of the equivalent stress in this area is far beyond the breaking resistance of the normal bone, but it should be mentioned that the surface on which it extends is a very small one, so we can say that in this area cracks of the iliac bone or fractures may occur are of small sizes.

Detail B shows the most affected area of the pelvic assembly as a result from impact. As we mentioned above the area was on the side part where the pelvic bone assembly took contact after the impact occurred. According to the scale of values the equivalent stresses presented in this area are about 304 [MPa]. These tensions spread on almost the entire upper surface of the iliac bone. In this area there will be definitely found fractures as a result of the impact.

Figure 3. The scale of equivalent stress

Figure 4. Frontal view of the stress distribution in normal pelvic bone assembly structure

Changing the mechanical properties of the bone and using the same scenario, we performed the finite element analyses of the osteoporotic pelvic bone assembly. The results of this analysis are presented using the images below. Figure 5 illustrates the scale of values of equivalent stresses
resulted from finite element analyses. The value 1.8 [MPa] represents the lowest limit of this scale, for which we chose the blue color. The highest value on this scale is 3,205 [MPa], red color being representative for this.

Helped by figure 6 we got details C and D which showed us the risk areas of the osteoporosis pelvic bone assembly.

Detail C shows the area where the impact between the vehicle and the pelvic bone assembly occurred. In this area we found on a very small surface equivalent stress of approximately 2,137 [MPa]. The values of equivalent stress being on a small area may cause cracks or fractures of small sizes in the iliac bone. The bone is the first part of pelvic assembly which takes contact with the vehicle.

Detail D shows the area that suffers the most from the impact. This is the area where the pelvic assembly is hit. The values of the equivalent stress are far beyond the braking limit for osteoporotic bone, at some points reaching even 714 [MPa]. Due to the values of the resulting equivalent stress but also of the area on which this extends, we can conclude that in this area there will be a high risk of producing fractures.

![Figure 5. The scale of equivalent](image)

![Figure 6. Frontal view of the stress distribution in osteoporotic pelvic bone assembly structure](image)

In the finite element analyses of each type of pelvic bone assembly we took into consideration the time the impact occurs is 0.5 [s]. From the ANSYS program we took the maximum and minimum values for equivalent stress distribution. We introduced these values in EXCEL program and we got the next graphs which are depicted with the help of figures 7 and 8.
Helped by figure 7 we can observe how the maximum values of equivalent stress can vary in time in the entire structure of the pelvic bone assembly. In this graph we have the equivalent stress variation for the both types of bones.

Figure 8 shows how the minimum values of equivalent stress for the both types of bones can vary in time in the entire structure of the pelvic bone assembly.

4. Conclusions
Following the finite element analyses we demonstrate that when the velocity at the impact moment is 70 Km/h the both types of pelvic bone assemblies, normal and osteoporotic, suffer fractures at the iliac crest level.
The resulted values are higher than those found in a real case because the impact was considered to occur between the vehicle and the pelvic assembly. The muscular system, the cartilages and the skin which would attenuate the power of the impact these were not considered.

5. References
[1] Gaiginschi R 2009 Road accidentology course (București: Editura Tehnică)
[2] Gaiginschi R 2009 The reconstruction and the road accidents expertise (București: Editura Tehnică)
[3] Arkusz K and Klekiel T 2018 The influence of osteoporotic bone structures of the pelvic-hip complex on stress distribution under impact load Acta of Bioengineering and Biomechanics 20 pp 29-38
[4] The 3D scanned models 2020 from: https://grabcad.com/library?page=1&time=all_time&sort=recent&query=pelvis
[5] Carter D R and Sprengler D M 1978 Mechanical properties and composition of cortical bone Clinical Orthopaedics and Related Research 135 pp192-217
[6] Bluc D, Nguyen N D, Nguyen T V, Eisman J A, Center J R 2013 Compound risk of high mortality following osteoporotic fracture and re-fracture in elderly women and men Journal of Bone and Mineral Research 28 pp 24-30
[7] Gillian L S Soles and Tania A. Ferguson 2012 Fragility fracture of the pelvis Current Reviews in Musculoskeletal Medicine 5 pp 222-228