VIRTUAL MODELLING OF MOTORCYCLE SAFETY HELMETS: PRACTICAL PROBLEMS

AUTHORS:

Alessandro Cernicchi, Ugo Galvanetto and Lorenzo Iannucci
Imperial College London

CORRESPONDENCE:

Alessandro Cernicchi
Department of Aeronautics
Imperial College London
South Kensington, London SW7 2AZ, UK
Telephone +44 (0)20 759 45113
Email a.cernicchi@imperial.ac.uk

ABSTRACT:

Motorcycle helmets are safety devices that can be optimised to perform better in different impact configurations. It is not easy to determine the mechanical properties of a particular model of helmet which maximise its effectiveness in real impact conditions and it is not surprising that most helmet producers use empirical design procedures and assess the effectiveness of their products by carrying out numerous experimental impact tests. The application of advanced computational techniques (i.e. Finite Element Method) to the study of the mechanical behaviour of helmets has been conducted with a variable degree of success in several cases. The use of virtual methods clearly provides a superior flexibility during the design process due to the simplicity by which the model is modified and retested.

This work is an attempt to clarify modelling aspects encountered in helmet virtual testing, such as mesh dependency of the results, the influence of retention system on the response and composite shell modelling. A Finite Element model of a commercially available helmet has been developed and impact tested reproducing the test conditions prescribed by the ECE 22.05 standards. Particular emphasis has been given to the analysis of fibre reinforced plastic helmets, which are currently under further development, due to their superior performance. The results have been compared with experimental data and possible reasons for discrepancies have been analysed.

KEYWORDS:

Helmet, Drop testing, Crashworthiness, Motorbike.

INTRODUCTION

Motorcyclists are among the most vulnerable road users due to a series of characteristics that make them qualitatively different from other road users. In the last decades the European Community has promoted a series of measures in an attempt to increase road users' safety. This has led to the introduction of several safety devices that have contributed to the reduction of the number of casualties and injuries in road accidents. One of the most important steps in terms of lives saved throughout the years was the introduction of the motorcycle helmet, the use of which is now compulsory almost everywhere in the European Union.

From an engineering point of view a helmet is a safety device that can be optimised to better perform in different impact configurations. This optimisation process is complicated by many complexities that characterise a safety helmet, such as the material non-linearity and the complex geometry of the helmet. Hence, the determination of the mechanical properties that a particular model of helmet should have to maximise its effectiveness in real impact conditions are not simple to determine and it is not surprising that most helmet producers use empirical design procedures, assessing the effectiveness of their products by carrying out numerous experimental impact test. This is mainly due to the fact that mathematical models able to predict the response of the helmet have been so far difficult to develop and have shown little reliability.

The application of FE models to the study of helmet has been conducted with reasonable success in several cases [1-3]; however, some modelling aspects have not yet been addressed. In this work the dependency of the results on the mesh density, the influence of the chin strap on the impact response and the modelling of the composite shell using material models available in Ls-Dyna are investigated.

HELMET FE MODEL AND TESTS DESCRIPTION

The helmet examined in this paper is an open face commercial helmet produced by Dainese s.p.a. (fig. 1). A Finite Element model of this helmet was built and the set of drop impact tests prescribed by the ECE 22.05 [4] were simulated using the commercial finite element solver Ls-Dyna 971 [5]. The helmet FE model comprised 4 different parts: a composite shell, an expanded polystyrene (EPS) energy absorbing liner, a retention system and a headform. However, the real helmet also contains two other parts: a low density comfort liner and a visor. The influence of the comfort liner on the dynamic response to the impact was assumed negligible. Owing to its extremely low crushing load, this should crush almost completely without transmitting any considerable load to the headform, thus it was not included in the model. Moreover, the influence of the visor and its attaching system was also deemed not relevant and they were also not included. In order to generate the finite element mesh the geometries of the different parts were imported into a commercial mesh-generator (Hypermesh), so that it was possible to auto-generate meshes which faithfully match the complex geometry of outer shell and impact absorbing foam.

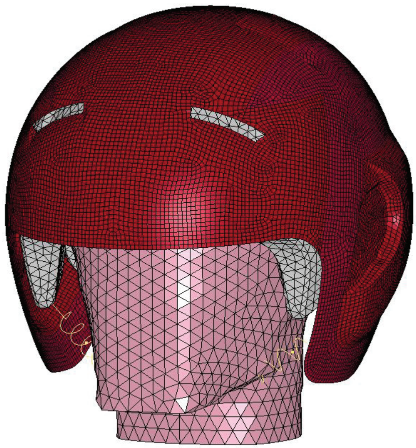

Figure 1: Helmet FE model.

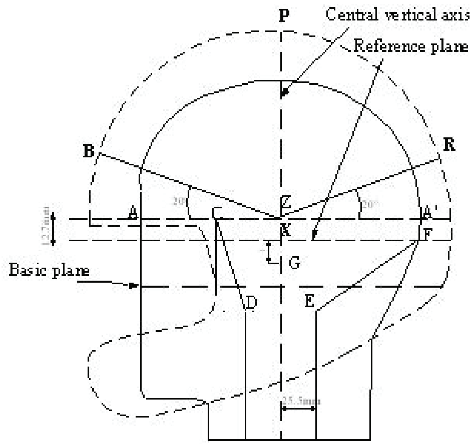

Figure 2: ECE22.05 points of impact.

ENERGY ABSORBING LINER MODELLING

The impact absorbing foam was modelled using four noded single integration tetrahedral elements due to the relative simplicity by which the mesh generator could handle the creation of good tetra-meshes even for the most complex geometries. However, tetrahedral element should be used with care: it is commonly agreed that cubic solid elements must be preferred since tetra meshes can sometimes lead to an excessive rigidity. This possibility was investigated along with the evaluation of the optimal mesh density (i.e. average number of element contained in a reference volume or area of the body meshed).

In order to evaluate possible differences between cubic and tetrahedral meshes the impact of a hemispherical indenter of 10 cm diameter to an EPS square mat of 100 cm^2 cross section and 2 cm thickness was simulated in Ls-Dyna. Six different meshes for the mat were tested: three cubic and three tetrahedral solid meshes of increasing density. In particular the coarse, medium and fine meshes had respectively 1, 2 and 4 elements through the thickness of the foam. The impact velocity and the mass of the indenter were set respectively to 6.7 m/s and 3 kg and the EPS mat was supported on its bottom surface. The indenter was modelled as a rigid body while the EPS was modelled using one of the material model available in Ls-Dyna: "crushable foam" [5]. This model allows the input of the material crushing stress against volumetric strain. The results in term of acceleration of the indenter during the impact for all the different meshes and an EPS density of 24 kg/m^3 are depicted in fig. 3.

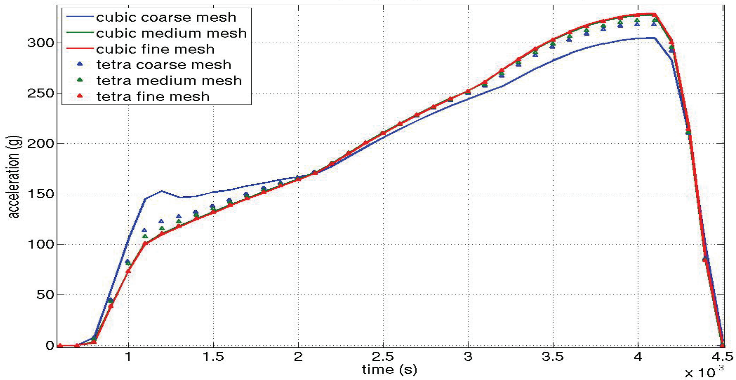

Figure 3: Indenter acceleration for different mesh densities and types of element.

As can be seen, there is complete convergence between the results of both tetrahedral and cubic fine meshes which proves that the use of tetrahedral elements is completely analogue to that of cubic ones for these particular case providing the mesh density is enough fine. Moreover, it can also be observed that there is good convergence between the results relative to medium and fine meshes, thus it was concluded that the element size used for the fine meshes are adequate to correctly simulate this class of problems. In the light of the results of these simulations, the helmet EPS foam was generated placing the same number of elements through the thickness as in the fine mesh (4-5 elements).

Another important aspect to be examined is the selection of an accurate load curve to represent the EPS behaviour. EPS belongs to the category of closed cell cellular solids. The classical shape of the compressive stress against compressive strain is characterized by three main regimes: the elastic zone for strain below 5%; the plateau zone in which the curve level out and a densification zone for high strains. Gibson and Ashby [6], using a simplified model for the single cells present in the material, gave an exhaustive physical description of the phenomena that lead to the determination of each zone. According to their model the three different zones of the compressive stress-strain curve of closed cell cellular solid can be inferred with the following equations:

$$\sigma = E\varepsilon + \frac{p_0 \varepsilon}{1 - \varepsilon - R} \qquad \text{Elastic} \qquad \text{(Eq. 1.a)}$$

$$\sigma = \sigma_y + \frac{p_0 \varepsilon}{1 - \varepsilon - R} \qquad \text{Plateau} \qquad \text{(Eq. 1.b)}$$

$$\sigma = \sigma_y \frac{1}{D}\left(\frac{\varepsilon_D}{\varepsilon_D - \varepsilon}\right)^m + \frac{p_0 \varepsilon}{1 - \varepsilon - R} \qquad \text{Densification} \qquad \text{(Eq. 1.c)}$$

For the definition of each parameter in the above equations the reader is referred to [6]. The data regarding EPS foams of different densities published by Di Landro et. al. [7] were collected and fitted with equations 1.a – 1.c using a least square method to derive all material constants. The knowledge of these material constants allows the determination of the compressive stress-strain curve for EPS given its relative density and constitutes. This is especially important when using the model for optimisation of a design. In the present analysis they were used to evaluate the curves relative to the two different EPS densities present in the helmet, and inserted into the Ls-Dyna input deck.

COMPOSITE SHELL MODELLING

The outer shell was modelled using four noded single integration shell elements. Also in this case, the mesh density is an important parameter that must be checked, thus a convergence study of the shell mesh density was performed. Six different shell meshes of increasing density were generated. The average element side ranged between 15 mm for the coarse mesh and 2 mm for the fine mesh. A series of impacts at point B (fig. 2) on a kerbstone anvil [4] were simulated in order to assess the optimal mesh density. Fig. 4 shows the contact force between shell and anvil for all mesh densities analysed. As can be observed, the convergence is reached only between meshes of average side of 2 and 3 mm (dashed lines), which indicates that the maximum level of accuracy of the solution can be achieved by using element of average side of 3 mm at most. These conclusions are based on the standard Ls-Dyna code.

It was then necessary to choose realistic material models for the shell elements. No experimental data about stiffnesses and elastic moduli were available for the composite materials present in the shell. Hence, an estimation of all the different parameter to be input in the material model was necessary. The helmet manufacturer used several types of woven and unidirectional reinforcements to accomplish the appropriate mechanical properties required. In particular, the reinforcements used are glass, carbon and Kevlar fibres, while the matrix material is vinylester. Among all the material models available in Ls-Dyna which can be used to represent composite materials with damage, material model 58 "laminated composite fabric" [5] appeared the most appropriate for the analyses. This material model takes into account the progressive failure of each composite layer within the shell element adopting a continuum damage mechanics approach. The components of the in-plane stiffness matrix are degraded by means of three damage variable that represent respectively the damage along the two principal material directions and shear damage. The damage variable evolution is implemented in the code and the user has no direct control on it. The input parameters required by the model are the stiffness matrix components, the maximum compressive, tensile and shear strengths and the maximum compressive, tensile and shear strains. All the elastic properties of matrix material and fibres were known. Hence, it was possible to estimate the elastic properties of each lamina in the model using the well-known Halpin-Tsai equations [8].

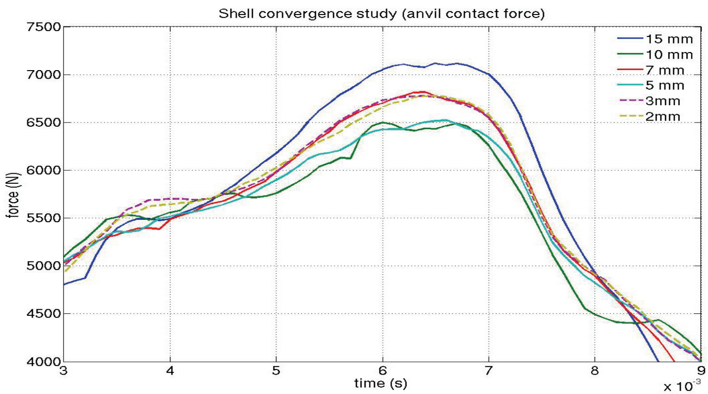

Figure 4: Shell convergence study anvil contact forces

Halpin-Tai equations allow the determination of elastic moduli; however, they do not give any information about final strengths and strains. The mechanisms which cause failure of a composite reinforced plastic laminate are extremely complex, and are not easy to determine knowing the fibre and matrix properties only. Many analytical and computational models have been proposed in literature. Although some of them have led to good results, in most cases the use of these advanced approaches has serious practical limits due to the fact that a large amount parameters not trivial to obtain are necessary for the models to be applied (local moduli of fibre and matrix, fibre and matrix, strengths, strength of the bond between fibre and matrix, reinforcement geometry etc…). Due to the lack of detailed data for the strengths of the matrix and fibres present in the helmet under examination the use of any of these advanced approaches would have been inaccurate. Therefore a very simplified approach was adopted:

- Under longitudinal tensile loads (along the fibre direction) each composite layer was assumed to fail when the fibre reached their ultimate tensile stress. Moreover, the ultimate tensile strain of the laminate was presumed to be equal to the fibre ultimate strain.

- Under transverse tensile loads (along the matrix direction) the composite strength was assumed to be controlled by the matrix ultimate strength and the material linear up to failure.
- Under longitudinal compressive loads the composite was assumed to fail when the transverse maximum tensile strain of the matrix was reached owing to Poisson's effect.
- Under transverse compressive loads the composite was assumed to fail when the matrix ultimate compressive strength was reached.
- Under shear loads the composite was assumed to yield when the matrix maximum shear strain was reached, the maximum shear strength was set to a high value according to experimental observation of analogue composites.

The strengths values derived using this approach were within the range of variation found in the open literature.

HEADFORM AND RETENTION SYSTEM MODELLING

The headform included in the model was generated according to the dimensions specified by the ECE 22.05. This component was modelled using 4-noded tetrahedral elements. Since the elastic properties of the headform are orders of magnitude higher than those of the other parts in the model, a rigid material model was used in FE model. The inertial properties of the headform where also set according to the ECE 22.05.

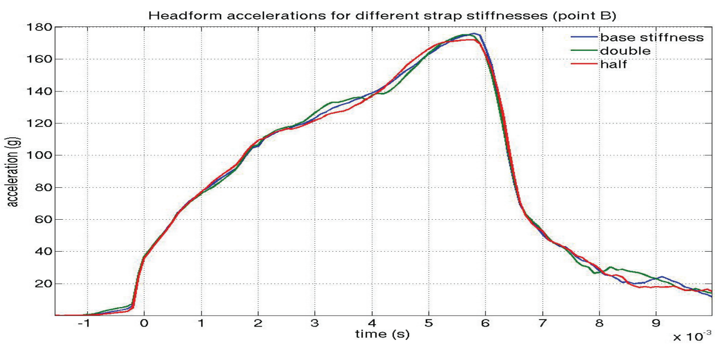

Figure 5: Influence of the chin strap stiffness on the headform acceleration for impacts on point B.

The retention system was modelled by two non-linear springs that linked the chin of the headform with the chin strap bolting points on the shell. The stiffness of the springs was set to zero under compression (a strap cannot sustain any compressive load) and to the value $4 \cdot 10^5$ N/m under tension loads. No experimental data were available for the strap used in the actual helmet. The stiffness was chosen so as to match a typical value for nylon straps of the same dimensions. The derivation of this stiffness is very important to correctly simulate the response of the system. However, a set of comparative numerical analyses of impact at points B and X with different chin strap stiffnesses revealed that this parameter does not actually influence the response of the system to a large extent (fig. 5). In the case of impact at point X, due to the inertial forces that act during the impact, the two attachment point of the chin-strap on the shell move towards each other. Hence, during the impact the chin strap is slung under the chin of the headform and does not affect its motion. Furthermore, the influence of the chin-strap on the impact response at points P and R is expected to be even smaller and thus was not investigated.

COMPARISON WITH EXPERIMENTS

Figs 7 and 8 show a comparison between experimental and numerical acceleration modulus recorded at the centre of gravity of the head form for impacts at point X. As can be observed, there is a good agreement between experimental and numerical results and all the discrepancies are within the range of the approximations made within the simulation. Acceptable agreements were also obtained for impacts at the other points that are not included for brevity. However, some other issues have to be addressed before helmet virtual modelling can be used reliably during the helmet design process.

The first aspect to be investigated is the influence of the temperature of the helmet response. In particular, the helmet was tested after being conditioned at -20 °C for the impacts on flat anvil and at +50 °C for the impacts on the flat one. This was taken into account in the numerical model degrading the mechanicals properties of the foam linearly with the temperature. However, this is not strictly correct and further investigations about the dependence of the material mechanical properties on the temperature are necessary if the accuracy of the results is to be improved.

A second important aspect to be discussed is the possible strain-rate dependence of the material properties. Many studies on composite materials have revealed a strain-rate enhancement which may need to be included to improve accuracy. Foam materials are also sensitive to variation of strain-rate. The strain rate enhancement was not included in the FE model. Ongoing experimental tests on EPS and various composite materials used in helmet manufacturing may shed light on material rate and temperature effects.

The last aspect to analyse is the suitability of the material model used for this particular problem. As specified earlier, the user cannot control the evolution of the damage variable. Furthermore, the model makes no differentiation between damage in the matrix and damage in the fibre. Experimental observation on impacted helmets showed that the main failure mechanism is matrix cracking. The onset of this type of damage is generally observed for loads of lower value than those necessary to rupture the fibres, thus it is required a separate damage variable to account for the degradation of the mechanical properties due to matrix cracking. The use of separate damage variables to model matrix cracking would improve the accuracy of the results obtained. The material model adopted to represent EPS assumes that the material crushes unidirectionally with almost no Poisson's effect. Moreover, the material assumes no interaction between the three main stress components that crush following the stress strain loading curve independently. All these hypotheses seem reasonable for EPS, but the implementation of a yielding criterion that better represent the tensile and compressive behaviour of EPS might bring about further accuracy.

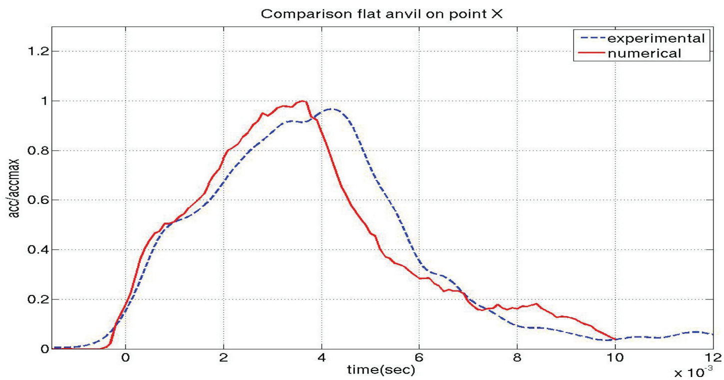

Figure 7: Comparison between experimental (blue line) and numerical (red line) headform accelerations, point X, flat anvil.

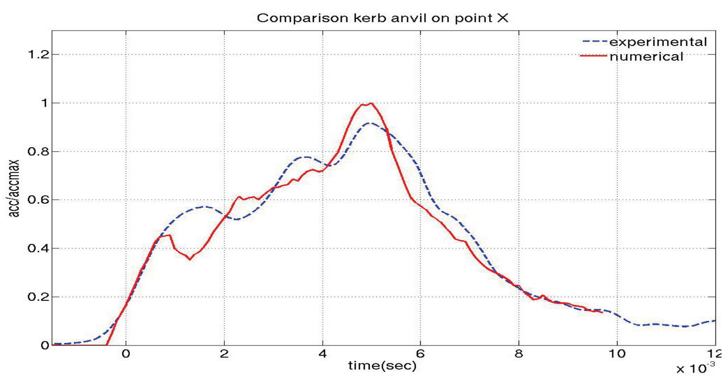

Figure 8: Comparison between experimental (blue line) and numerical (red line) headform accelerations, point X, kerbstone anvil.

CONCLUSIONS

A simplified procedure for the determination of the mechanical properties of the composite shell based on the Halpin-Tsai equations was adopted. Some important aspect such as the choice of optimal mesh density for the different components of the model and the influence of chin strap on the impact response were addressed. The FE model was used to run a full set of simulations representing the drop tests specified by the ECE 22.05 and the numerical results were compared with experimental data. Satisfactory agreement was observed and all the possible causes for the discrepancies noticed were identified in order to suggest possible improvement on the helmet FE model. The implementation of energy based fracture models, including rate enhancements, for intralaminar damage within Ls-Dyna should further improve the correlations with tests.

REFERENCES

[1] D.S. Liu, C.M. Fan, M.C. Lee, C.Y. Yen. "A Dynamic Finite Element Simulation of the Impact of Motorcycle Helmets", Crashworthiness, Occupant Protection and Biomechanics in Transportation Systems, 38, pp. 199-206, 1997.

[2] V. Kostopoulos, Y.P. Markopoulos, G. Giannopoulos, D.E. Vlachos. "Finite Element Analysis of Impact Damage Response of Composite Motorcycle Safety Helmet", Composites: Part B, 33, pp. 99-107, 2002.

[3] F.M. Shuaeib, A.M.S. Hamouda, M.M. Hamdan, R.S. Radin Umar, M.S.J. Hashmi. "Motorcycle Helmet. Part I Biomechanics and Computational Issues", Journal of Materials Processing and Technology, 123, pp.406-421, 2002.

[4] United Nations. "Uniform Provision Concerning the Approval of Protective Helmets and Their Visors for Driver and Passengers of Motor Cycles and Mopeds", ECE Regulation 22.05, 2002.

[5] J.O. Hallquist. "Ls-Dyna Keyword User's Manual – Version 971", Livermore Software Technology Corporation, 2006.

[6] L.J. Gibson, M.F. Ashby. "Cellular Solids. Structure and Properties – Second Edition", Cambridge, 2001.

[7] L. Di Landro, G. Sala, D. Olivieri. "Deformation Mechanisms and Energy Absorption of Polystyrene Foams for Protective Helmets", Polymer Testing, 21, pp. 217-228, 2002.

[8] J.C. Halpin; "Primer on Composite Materials Analysis", CRC Press, 1992.