Development of 3D finite element model of umbilical systems for offshore application

Dr Michaël Martinez, Timothée Perdrizet
IFP, Applied Mechanics Division, BP n°3, 69390 Vernaison, France

Ian Probyn
DUCO Ltd, Walker Riverside, Newcastle upon Tyne, UK

Abstract : Umbilical systems provide power and control (electricity, hydraulic power, chemical injection) to subsea oil and gas equipment. Electrical cables and hydraulic tubes are arranged in helical bundles and placed in successive layers depending on the design. Installations into increasingly deeper water place greater demands upon the structural components in terms of strength. 3D finite element analysis is an applicable technique in which to assess the strain and stress distribution and the interactions between components in such complex structures. The true geometry is respected and most of the simplifications of usual approaches can be relieved. However finite element models developed this way are complex to build and can reach a large size, the model construction and the simulation must be adapted in order to remain practical. Abaqus/Explicit is used as a solver because it can deal with large size problems and numerous contact conditions which existing in umbilicals. Quasi-static analysis techniques have been developed to simulate the umbilical behaviour in tension, bending and crushing load cases and are validated against available test data.

Keywords : umbilical, offshore, complex structures, Abaqus/Explicit, parallel execution

1 Introduction

Subsea umbilical systems provide the vital supply and control link from platforms or topside vessels to subsea oil and gas equipment. Typical water depth for an umbilical is about 2km, but future applications are ever increasing and 3km will soon be exceeded. The length of an umbilical is dependent upon its function and field layout, some have been manufactured in a continuous length as long as 200km.

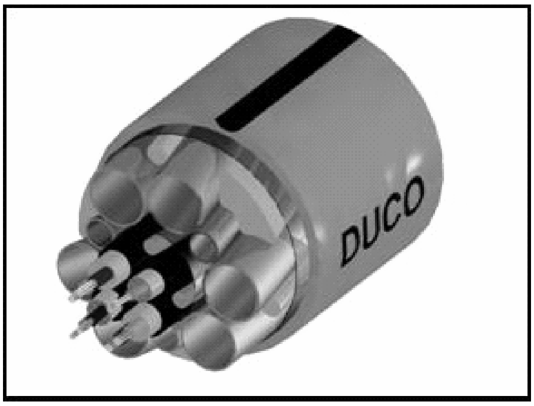

Figure 1. Typical steel tube umbilical.

There are several different types of umbilical and the requirements of the particular installation will dictate which is most suitable. However this paper will concentrate on steel tube umbilicals which are typically used for deeper water applications. A steel tube umbilical consists of steel tube fluid conduits, electrical cables and fibre optic cables arranged in contra-helically wound layers as shown in Figure 1. The bundle is contained by binding tape providing radial reinforcement and protected by a polymer outer sheath. The primary function is to provide :

- chemical injection for flow assurance,
- electrical signals for valve control and monitoring,
- hydraulic pressure for valve actuation,
- electrical power for subsea pumping,
- fibre optics for data acquisition and monitoring

An umbilical will be designed for either static or dynamic service. Both experience high tensile load during installation, but once a static umbilical has been laid, or trenched in the seabed, the tubes are only subjected to internal pressure and hydrostatic loading. Dynamic umbilicals are designed to hang in a catenary from a floating vessel and are subjected to high tensile loading and fatigue mechanisms due to the motion of the vessel. The steel tubes provide the main tensile strength but must also contain high internal pressure. The dimensions of the tubes are driven by the flow rate requirement, which determines the bore size, and the pressure and tensile capacity, which determine the wall thickness. The structural behaviour is further complicated because the bending stiffness of the umbilical varies according to applied tension and friction between the layers.

Traditional methods used to assess the suitability of umbilicals are based on analytical formula which are partly empirical and present a high degree of conservatism. Two dimensional FE methods are also used, but a high level of interpretation is needed when transferring these results to three dimensions. Three dimensional FE methods have not been developed before, because they have been considered prohibitive in terms of development difficulties and simulation time. However, they respect true geometry, accurately assess the stress and strain distribution and the interaction between components. Most of the simplifications of traditional analytical approaches can be relieved. The objective of this work is to prove that such methods are applicable and that the results they provide are valid.

2 Umbilical design and loading

An umbilical design was selected for the study (figure 2). It consisted of a central tube surrounded by a bundle of steel hydraulic lines and electrical cables. The presence of a large diameter central tube is not conventional in an umbilical and DUCO was interested in a detailed analysis of the behaviour of such a structure. Experimental test results exist and were used to prove the validity of the FE model. The results of the FE simulations were also compared to existing analytical formula where it was possible. Tensile, bending and crushing tests were simulated as closely as possible to the experiments. The simulations were performed with and without pressure in the hydraulic lines. Comparison between simulation and experimentation was performed on the bending and crushing tests only, as the experimental tensile test results appeared to be questionable on this particular case. However the tensile test presented little uncertainty from a FE modelling point of view and comparison with analytical results was sufficient. The bending test dealt with the bending stiffness of the umbilical, which was dominated by the steel tubes and, if any, the friction between them. The crushing test dealt with the radial stiffness of the umbilical which was dominated by the softer components: outer sheath, coatings and electrical cables. These two tests showed a good comparison and provided a good validation frame for the FE model.

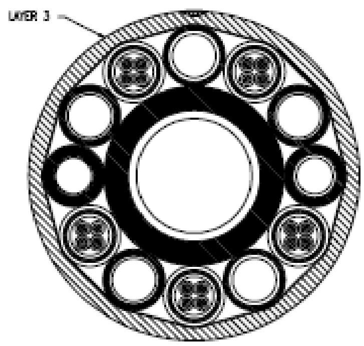

Figure 2. Umbilical design selected for the three dimensional FEA.

3 FE model

The objective of the three dimensional FE modelling was to obtain a precise distribution of the strains and stresses within the structure. Subsequently the true geometry must be respected as much as possible and therefore each component was modelled individually. The components are made of very different materials : polymers of various grade are used for the fillers, outer sheath and coatings, stainless steel is used for the hydraulic lines, the binding tape is made of fibre reinforced polymer. The electrical cables are complex structures : copper and polymer are used for the conductors and coatings, high strength steel is used for the armour. The properties of these materials are very different; the Young's modulus of steel is around one thousand times that of the polymer. So the behaviour of the component is highly dependent on its constitutive materials and the way they are arranged. Depending on the loading, the behaviour is dominated by one type of material or another. Tension and bending behaviour is driven by the steel components whilst the behaviour in radial crushing is, at first, given by the polymeric components. Different material models from Abaqus are used in the FE model : isotropic elasto-plasticity, orthotropic elasticity, depending on the material and on the loading. Material Property values were identified from the DUCO data base.

Meshing is a key point of the model. The mesh must be quite refined, because each component is modelled individually. However its density must be reasonable so that the model size and solver time remain acceptable. A careful choice of element type helps to contain the mesh within convenient limits. Shell elements are utilised for thin structures such as the steel tubes. They provide, at a much lower calculation time, a better bending behaviour than solid elements. However they present a strong limitation for our application : the contact surface is not properly defined when the ratio between the thickness and other dimensions is too high. Where it is not acceptable, solid elements are used instead. Beam elements are used to model the steel wires of the electrical cables armour. They are convenient as long as the contact between wires is negligible. The tapes are modelled with membrane elements : They provide the required tensile stiffness and a null bending stiffness. Solid elements are used whenever structural elements are not convenient. Their main advantage is to provide a good geometrical description of the contact provided the mesh is refined enough. Their drawback is the superior mesh refinement they require. Because a correct contact description is mandatory for our simulations, most of the components are actually modelled with solid elements and the model size reaches quite high values : up to two million of degrees of freedom in some applications.

Abaqus/Explicit is used to run the models and it is well adapted to solve large size models with numerous contact conditions. Abaqus/Explicit solves directly, without iteration, according to an explicit scheme based upon the dynamic equation. No tangent matrix has to be stored and the memory required by the simulation is limited, even for large size models. The convergence is not checked but the small size of the time increment, along with the stability of the dynamic equation, guarantees reliable results provided some additional checks are made by the user. Even though Abaqus/Explicit is a dynamic solver it can be used to solve a static problem through using a quasi-static approach. The equilibrium state is achieved if the dynamic force is negligible. For such applications, the simulation time has no physical meaning and could be tuned for convenience, provided the viscosity of the materials is negligible. Therefore to minimise the dynamic force and the simulation time the velocity and acceleration have to be controlled. The quasi-static assumption is checked by determining the ratio between internal and kinetic energy. The kinetic energy must remain significantly smaller than the internal energy. If this is not the case, the dynamic effect is not negligible and the quasi-static assumption is not valid. An assessment is made to ensure that the kinetic energy remains stable and smooth, and that the velocity of some specific points of the structure is as expected. The load is always applied in a smooth and steady manner to promote quasi-static behaviour and the structure should respond accordingly. If the kinetic energy presents a large variation, the velocity of some part of the structure is not steady and part of the problem is probably ill-conditioned. The time increment size is imposed by stability conditions and is generally very small. As a consequence the number of increments needed to complete a step is very large. The simulations presented hereafter were achieved in 200 000 to 500 000 increments.

4 Parallel execution

The large problem size and the large number of time increments lead to time consuming simulations. To speed-up the simulation parallel execution was used. Parallel execution is quite efficient in Abaqus/Explicit and allows quite heavy simulations to be performed in a reasonable time period. Table 1 presents a test made on a medium scale model. The test was performed on a linux workstation with Intel Xeon 3 GHz (one processor) and on a supercomputer cluster with AMD Opteron 2.2 GHz processors. The test on a single processor of the cluster was to quantify the improvement due to the processor only, and then the test was repeated on four processors. Even running on one processor, the PC cluster was already more efficient than the workstation, but a much larger gain was obtained by running on four processors. The gain decreased with the size of the model, but even on a large size model (one of those presented in this study) the speed-up factor remained greater than the number of processors (table 2).

Processor	Number of CPU	Solver time	Speedup factor
Intel Xeon 3 GHz	1	55 min	1
AMD Opteron 2.2 GHz	1	42 min	1.3
AMD Opteron 2.2 GHz	4	12 min	4.58

Table 1. Solver time for different computer configurations, medium scale model.

Processor	Number of CPU	Solver time	Speedup factor
Intel Xeon 3 GHz	1	104 h	1
AMD Opteron 2.2 GHz	4	24 h	4.33

Table 2. Solver time for two computer configurations, large scale model.

5 Tensile test

One pitch of the umbilical, 2.5 m long, was modelled in Abaqus. Kinematic couplings were applied at both end sections. The simulation was performed with pressure in the tubes. The pressure was the service pressure which ranged from 5 000 psi (34.47 MPa) to 10 000 psi (68.95 MPa) depending on the tube. The pressure added an internal tension to the umbilical (end cap effect). Figure 3 presents the applied tension, which did not include the end cap effect. The axial deformation was the global umbilical deformation. The curve provides the axial stiffness K_{axial}^{num} and the yield load T_{yield}^{num} of the umbilical.

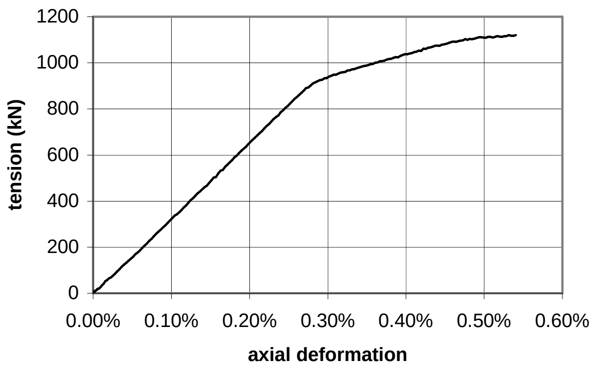

Figure 3. Axial tension applied to the umbilical, tensile test, FE simulation.

The numerical characteristics are very close to those provided by a simple mixture law. The mixture law includes the helical shape of the peripheral components :

$$K_{axial}^{anal} = \sum_{comp} S_{comp} E_{comp} \cos^2 \alpha_{comp}$$

$$T_{yield}^{anal} = \sum_{comp} S_{comp} \sigma_y^{comp}$$

Where S_{comp} is the cross-section surface of the component, α_{comp} its lay angle, E_{comp} and σ_y^{comp} the Young modulus and yield stress of its constitutive material. Table 3 provides the ratio between the numerical and analytical stiffness and yield loads. The numerical stiffness is slightly lower than the one given by the mixture law. The difference in yield load is more important, but the assumption of mixture law is less accurate. It supposed that all the components yield at the same deformation whilst in reality, and in the FE model as well, the components yield at different deformations, depending on their own stiffness and yield load. The numerical yield load accounted for the first yield. Moreover the inner pressure added extra tensile stress, both in hoop and axial directions, which reduced the yield load. The mixture law does not take this into account.

$K_{axial}^{num} / K_{axia}^{anal}$	0.96
$T_{yield}^{num} / T_{yield}^{anal}$	0.82

Table 3. Ratio between the axial stiffness and yield load determined from FE simulation and from analytical estimation. Tensile test.

The FE simulation provided the stress distribution in the steel tubes. Figure 4 shows a slice of the umbilical where only the steel tubes are plotted, so that the stress distribution can be seen. One must imagine that these tubes are actually surrounded by their coatings, the electrical cables and the outer sheath. It can be seen that the axial stress is not homogeneous, it is indeed related to the hoop stress. The hoop stress is attributed to, firstly for the inner pressure and secondly for the tightening of the outer tubes around the inner tube. This tightening tends to ovalise the tubes. Figure 5 shows the evolution of the axial and hoop stress on the inner and outer surfaces of a peripheral tube. Pressurisation is applied before the tension is applied, thus the initial value of axial and hoop stresses are non null. Axial stress increases as tension is applied and the yield load appears clearly. Hoop stress remains more or less constant and an ovalisation load slightly increases the gradient between inner and outer surfaces.

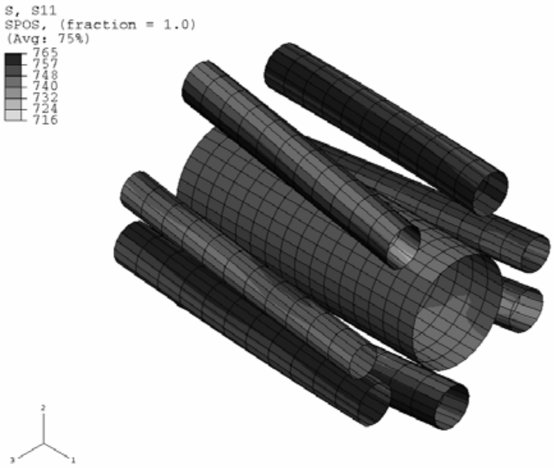

Figure 4. Tensile test : axial stress in the steel tubes.

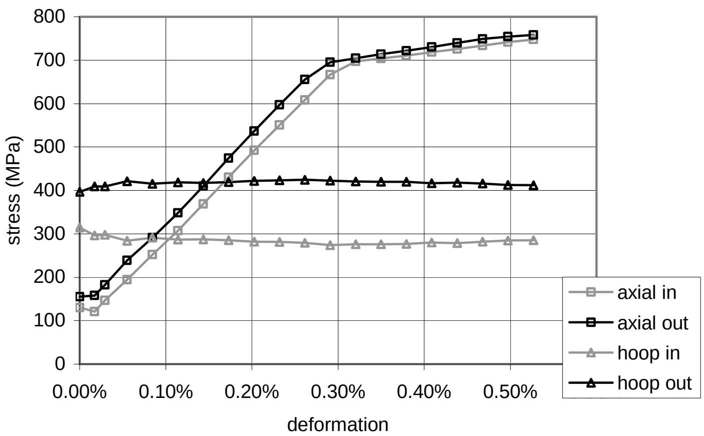

Figure 5. Axial and hoop stress at the inner and outer surfaces of a peripheral steel tube. Tensile test.

6 Three point bending test

The umbilical was held at both of its extremities by two support blocks. A tension strap pulled the umbilical at its centre (figure 6). The pull load and the umbilical curvature near the pull strap were monitored. The curvature was given by the relative displacement of three points marked on the umbilical. The experiment was performed on a pressurized umbilical, at the service pressure, from 5 000 psi (34.47 MPa) to 10 000 psi (68.95 MPa) depending on the tube. The Abaqus model was an exact representation of the test; the support blocks and the pull strap were modelled as analytical rigid surfaces (figure 7). Loading was applied via the displacement of the pull strap.

Figure 6. DUCO three point bending test rig.

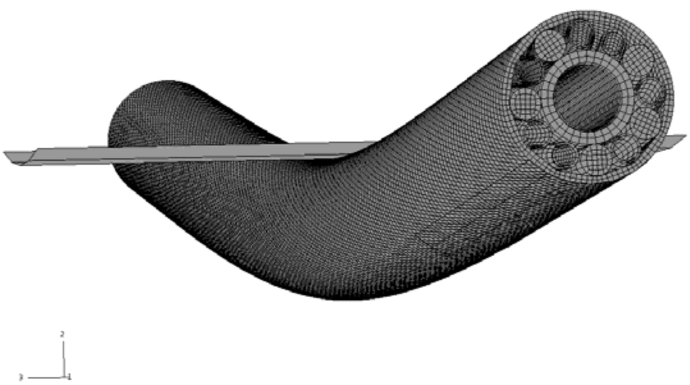

Figure 7. Three point bending simulation in Abaqus/Explicit.

The figure 8 shows the bending moment versus the curvature for both the FE simulations and the experiment. The monitoring of the curvature was performed in exactly the same manner in the simulation as in the experiment. The experimental and FE curves are quite close, which proves that both the simulation and the test are reliable. The traditional way to calculate the bending stiffness of such a structure is to consider each component bending individually according to the global curvature. In doing so the friction between the components is neglected. The bending stiffness is then given by :

$$K_{bend}^{anal} = \sum_c \frac{2 \cos \alpha_c}{\frac{1}{E_c I_c} + \frac{\cos^2 \alpha_c}{E_c I_c} + \frac{\sin^2 \alpha_c}{2\, G_c I_c}}$$

Where I_c, α_c are the inertia and lay angle of the component, E_c, G_c the Young and shear modulus of its constitutive material. Table 4 provides the ratio between the numerical and experimental stiffness, and the ratio between the numerical and analytical stiffness. The first ratio is reasonably close to one, if we consider the difficulty to get such stiffness from full scale experimental test. The second ratio is also very close to one. This means that friction between the components is negligible in comparison to the bending load for this umbilical design.

$K_{bend}^{num} / K_{bend}^{exp}$	1.22
$K_{bend}^{num} / K_{bend}^{anal}$	1.02

Table 4. Ratio between the numerical, experimental and analytical bending stiffness. Bending test.

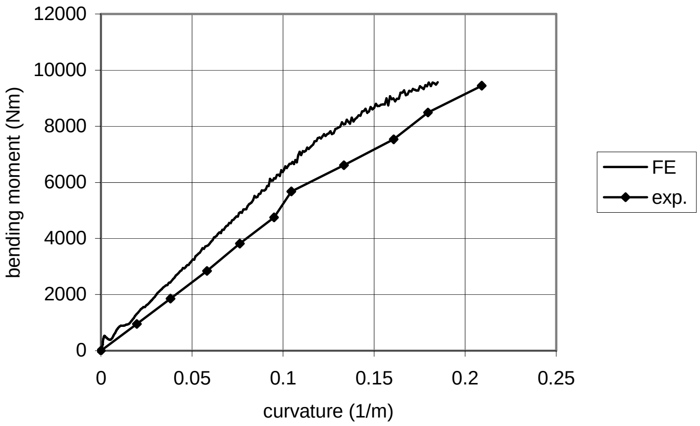

Figure 8. Bending moment at the umbilical centre. Comparison between the FE simulations (FE) and the experimental curve (exp). Bending test.

Figure 9 shows the axial stress distribution in the steel tubes at the umbilical centre. As for the tensile test, one must imagine that these tubes are actually surrounded by their coatings, the electrical cables and the outer sheath. Bending is equally distributed to the tubes, the stress level varies according to the tube diameter. The tube in the top position is under the central former and there is a stress concentration directly under the contact zone. Figure 10 shows the stress evolution on a peripheral tube. The internal pressure is applied at the very beginning of the step and induces a rapid increase of the hoop and axial (end cap effect) stress. The axial stress then increases linearly according to the curvature. The gradient in hoop stress between the inner and outer surfaces increases because of the ovalisation load.

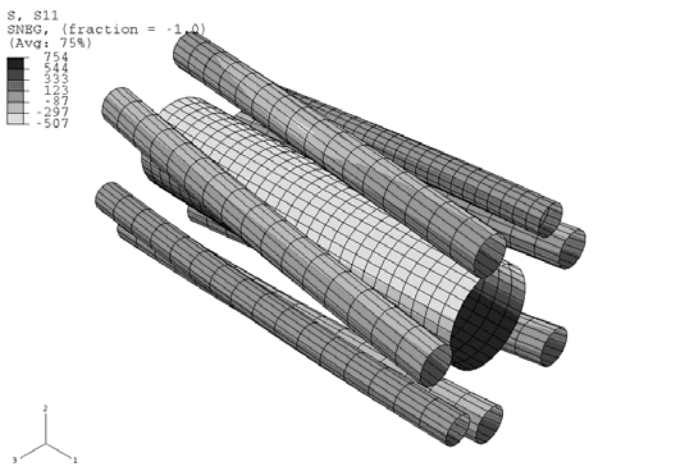

Figure 9. Axial stress distribution in the steel tubes at the umbilical centre. Bending test.

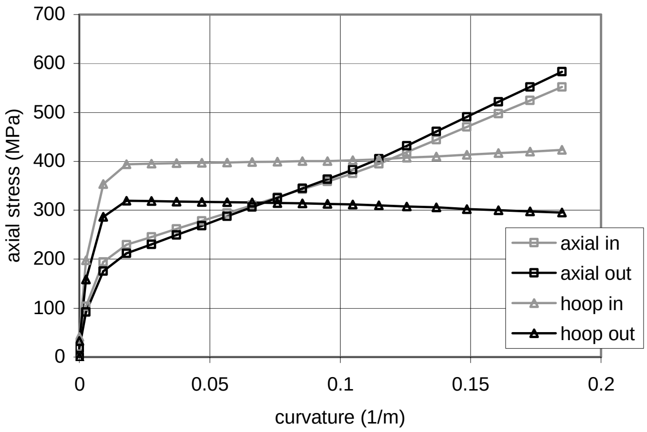

Figure 10. Axial and hoop stress at the inner and outer surfaces of a peripheral steel tube. Bending test.

7 Crushing test

The umbilical was pressed between two pads (figure 11). The load on the pads and their displacement were monitored. The test was performed without pressure in the umbilical. The simulation is an exact representation of the experiment (figure 12). The two pads are modelled as rigid surfaces and their velocity is imposed. No boundary conditions are applied to the umbilical itself. The FE model of the crushing test is rather different from the one of the tensile and bending tests. The loading is localised, there is no need to model a full pitch length of umbilical, but the radial behaviour needs to be a very precise description. All components, including the tubes, receive a fine solid mesh. This gives access to the radial behaviour of the tubes and improves the contact definition between tubes and coatings. On the whole, the problem size is quite similar to the tensile and bending tests.

Figure 11. DUCO crushing test.

Figure 12. FE model of the crushing test.

Figure 13 presents the crush load evolution for the experiment and the simulation. The position 0° and 45° refer to two different orientations of the umbilical between the pads. The fit is quite good between the experiment and the simulation. Figure 14 plots the minimal principal strain centre at the umbilical centre. This is the highest strain in magnitude since the umbilical is loaded in compression. The strain is localised and only a fine mesh could represent it. The strain is important in the outer sheath, electrical cables and coatings placed between the two pads. In comparison the steel tubes did not deform and acted as rigid bodies. The crushing behaviour, at least for this load range, is driven by the behaviour of the outer sheath, electrical cables and coatings.

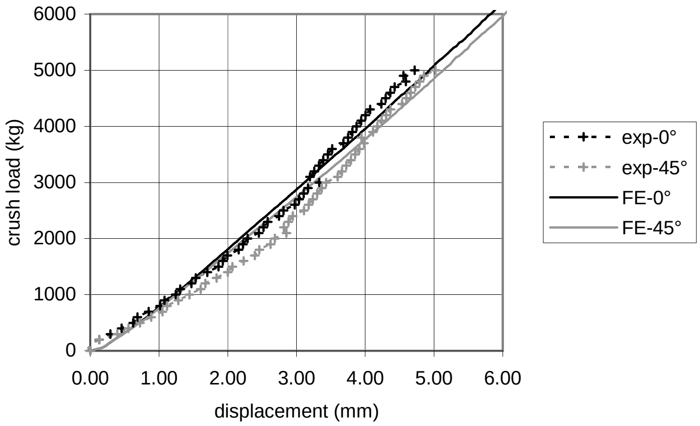

Figure 13. Crush load, comparison between the experimental test and the FE simulation for two positions of the umbilical on the rig.

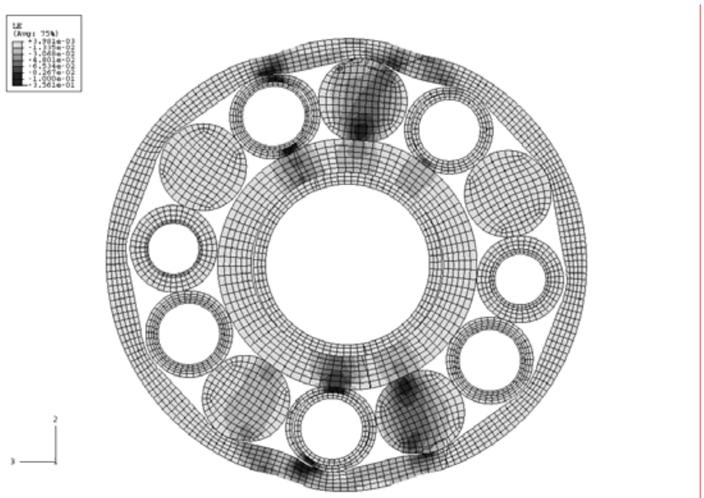

Figure 14. Minimal (compression) principal strain and the centre of the umbilical at the end of the crushing test.

8 Armoured electrical cable modelling

The electrical cable required a separate model to determine an equivalent representation for inclusion in the full umbilical FE model. Indeed, its structure is quite complicated : coatings, helically wrapped electrical conductors, fillers, and armour steel wires (figure 15). The armour provides high axial stiffness to the cable without greatly increasing its bending stiffness but plays a minor role in the crushing stiffness. Thus the behaviour of the cable is highly anisotropic and its estimation is not straightforward.

An FE model of the electrical cable alone was developed. This model was quite detailed, the electrical conductors and armour wires were individually modelled (figure 16). A very similar study to the one performed on the umbilical was performed on the electrical cable : tensile, bending and crushing tests were simulated. Yet there was no experimental test data to compare with, but comparisons were made with analytical formulas for the tensile and bending tests. No analytical formula was available for the crushing behaviour. The comparison of the stiffness is presented in table 5, the analytical stiffness were calculated as for the umbilical.

$K_{axial}^{num} / K_{axial}^{anal}$	0.94
$K_{bend}^{num} / K_{bend}^{anal}$	1.21

Table 5. Ratio between the numerical and analytical tensile and bending stiffness. Electrical cable model.

The axial stiffness ratio is similar to the one found for the whole umbilical (table 3). The analytical computation of bending stiffness neglects friction. But friction in the electric cable is not negligible because of the large number of contact surfaces (12 contact lines between the peripheral lines and the central tube in the umbilical, against 42 contact lines between the armour wires and the inner core). This is why the bending stiffness ratio is higher than one.

The behaviour of the electric cable and the umbilical are slightly different. In this umbilical, tensile and bending stiffness are dominated by the central tube. In the electrical cable, axial stiffness is given by the armours, bending stiffness is given by the inner core and the armours/inner core interaction (contact pressures, friction, and occurrence of sliding).

The electrical cable was modelled in the umbilical as a bulk homogeneous material. The tensile, bending and crushing tests performed on the detailed electrical cable model provide its tensile, bending and crushing stiffness. These stiffnesses were converted into equivalent Young's moduli and were used in the global umbilical model.

Figure 15. Armoured electrical cable used in the umbilical.

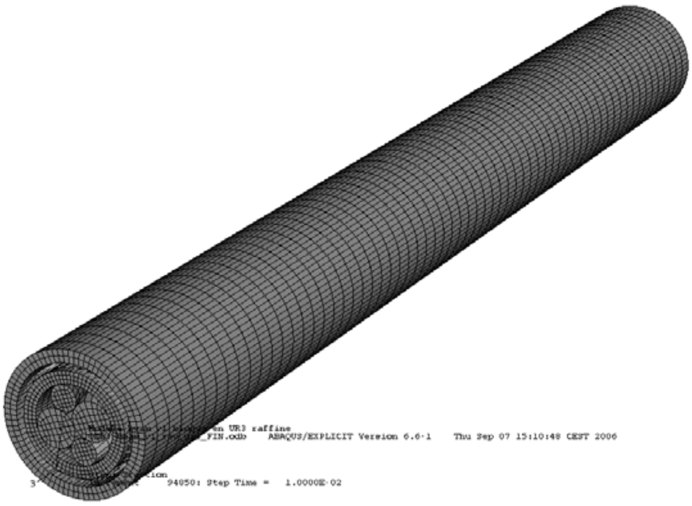

Figure 16. FE model of the armoured electrical cable.

9 Conclusion

The objective of the study was to prove that three dimensional FEA is a practical technique for modelling umbilicals. Detailed models of an umbilical, where all the components were individually accounted for, were developed. Such models respected the true geometry of the structure, they accurately assess the stress and strain distribution and the interaction between components. Most of the simplifications of traditional analytical approaches were relieved. As a consequence, the size of these models was quite large. However, thanks to the use of Abaqus/Explicit and to parallel execution, these models solved in a reasonable time period, around 24 hours. The computer resources remained reasonable since the simulations were performed on four processors and the memory requirement was quite small.

The FE simulations were validated against full-scale tests performed by DUCO : three point bending and radial crushing tests. These two tests were complementary because they loaded the umbilical in two very different ways. The experimental and numerical stiffness were quite close for both tests. Numerical stiffness was also in good agreement with analytical formula for tensile and bending tests. The differences that existed could easily be explained by the simplifications of the analytical approach. Once validated, the FE simulation provided a deep and detailed insight into understanding the umbilical behaviour. It was then possible to follow the evolution of the stress components at any point in the structure.

Umbilical modelling required a specific study for the armoured electrical cable. The behaviour of this component was not straightforward. A complete three dimensional model was constructed in a similar manner as for the umbilical, which allowed simplified characterization in tension, bending and radial crushing.