Stresses in armour layers of flexible pipes : comparison of Abaqus models

T. Perdrizet[1], J.M. Leroy[1], N. Barbin[1], V. Le-Corre[1], D. Charliac[2], P. Estrier[2]

[1] IFP Energies Nouvelles, [2] Technip

Abstract: The accurate modeling of offshore flexible risers behaviour remains a great challenge because of (i) their complex internal structure, (ii) the variable nature of the loads along the pipe (tension, curvature, internal and external pressures), (iii) and the interactions with structures used to limit the pipe curvature. Technip and IFP Energies Nouvelles have been codeveloping for many years models dedicated to stresses calculation in the armour wires, to assess the flexible lifetime. These models must account for a large number of potential inner contacts (contacts between upper and lower layers, lateral contacts between adjacent armour wires in the same layer) as well as external contacts (bend stiffener, arch, bellmouth or other curvature limitation setup). This paper presents implicit and explicit Abaqus simulations of a cyclic bending test of a flexible pipe with internal pressure and tension. The simulation results are successfully compared to experimental data. Moreover, the parallel performances of the explicit and implicit solvers are evaluated and highlight the efficiency of the Abaqus/Standard parallelisation up to 64 CPU cores for our applications.

Keywords: Abaqus, flexible pipe, bending stiffener, armour, kinematic, friction, implicit, explicit, parallel performance

1. Introduction

High pressure flexible risers have been used in offshore oil and gas industry for over 30 years. These pipes transfer produced fluids from the seabed to the surface facilities or transfer injection fluids, control fluids or lift gas from the surface facilities to the seabed.

A typical flexible riser, as described in Figure 1, is composed of thermoplastic sheaths and helically wrapped layers dedicated to specific functions depending on their laying angle:

- thermoplastic sheaths provide fluid transportation capacity (Pressure sheath) and/or pipe protection (External sheath),
- helically wrapped layers with high angle (short pitch) resist to hoop stresses due to internal and external pressures (Carcass, Pressure vault),
- helically wrapped layers with small angle (long pitch) support axial loads (Armours). These latter are composed of a minimum of 2 layers with opposite laying angles to equilibrate torsion torque, each one containing several (up to 100) armour wires,

- anti-wear thermoplastic layers separate steel layers to avoid steel to steel contacts.

However, each flexible riser is made fit-for-purpose: layers and materials are chosen depending on the required internal diameter, on internal fluid composition, pressure and temperature, of the offshore environment and water depth, on insulation needs, etc.

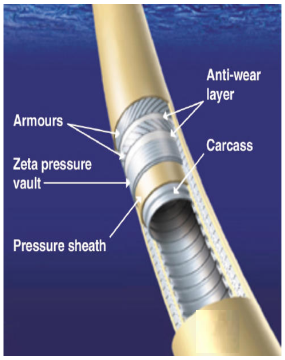

Figure 1. Typical flexible riser

Technip's flexible risers are unbounded, and flexibility in bending is obtained by relative slips between layers. This flexibility is used offshore to comply with the large movements of the topside structure, subjected to waves, currents and wind. Moreover, the flexibility allows the risers to be manufactured (and controlled) in continuous lengths onshore, bent on reels or baskets and quickly unreeled offshore for installation (**Figure 2**, **Figure 3**).

Figure 2. Pipelay vessel

Figure 3. Top of flexible risers

The accurate modelling of offshore flexible risers behaviour, necessary to assess the layers lifetime, remains a great challenge because of :

- their complex internal structure,

- the interaction between layers, including friction: a large number of inner contacts can occur (contacts between upper and lower layers, lateral contacts between adjacent armour wires in the same layer),
- the variable nature of the loads along the pipe (tension, curvature, internal and external pressures),
- the time dependent loads (tension and curvature depend on waves, currents and floating vessel response),
- the interactions with structures used to limit the pipe curvature,
- the service life time (typically 20 years, with a safety factor of 10).

This paper is focused on the stress assessment in the armours layers, which provide the pipe axial strength.

Technip and IFP have been codeveloping for many years models dedicated to stresses calculation in these armour wires, to assess the flexible lifetime. A comparison of these semi analytical and finite elements models was presented in (Leroy, 2010). In particular, a finite element simulation of a full length flexible riser, using the explicit integration scheme (Abaqus Explicit) and running on a parallel platform, was described and successfully compared to experimental test results.

Hereafter, we present an improvement of this model, using the implicit scheme (Abaqus Standard). The Implicit and Explicit models are used to simulate a riser tested in the Technip's flexible pipe R&D center. They include end fittings effects, interactions with a stiffener, internal pressure and tension loads and curvatures resulting from cyclic bending. The two models are specially compared in terms of accuracy, using strain measurements on the tested riser.

2. Case study

The studied pipe is a 8" internal diameter riser with two armours layers (laying angle: +/-30°) separated with an anti-wear tape. The total length is about 13 m, including the two end fittings. A stiffener was introduced for the test (**Figure 4**). The assembly is tested in a dedicated test rig (**Figure 5**, and **Figure 7**), where the pipe is submitted to internal pressure, axial tension (applied with a tension jack) and bending cycles (rotations of the swinging table, **Figure 6**). This loading is representative of a top of a riser, where rotations are governed by the topside structure.

Figure 4. ; The studied flexible riser with the stiffener

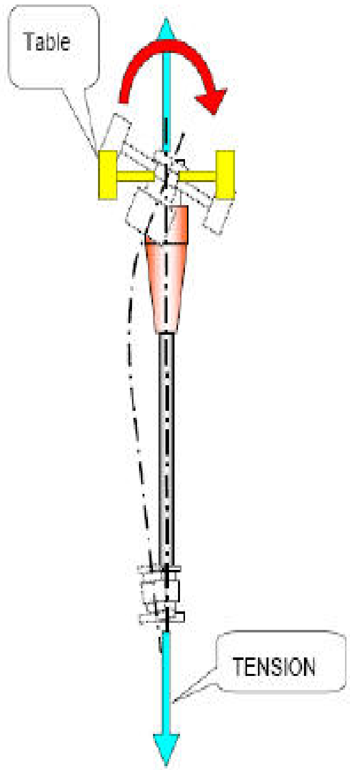

Figure 5. Schematization of the test rig

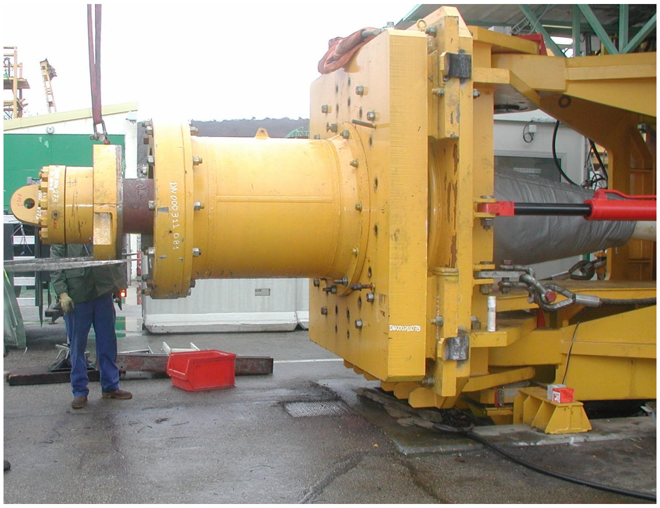

Figure 6. Introduction of the top part on the swinging table

Figure 7. Test rig overview

Some windows have been made in the external sheath to stick some strain gages on the external armour layer (**Figure 8**). Some of theses gages are two parallel strain gages on the same armour: the two strain measurements give both tangential and bending strains. **Figure 9** shows locations (red stars) of these windows along the pipe: 7 sections were instrumented, specially under the stiffener.

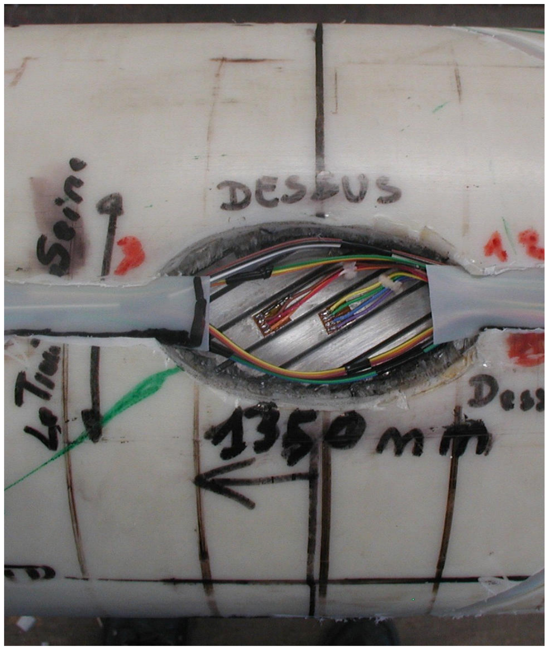

Figure 8. Example of strain gages on external armours.

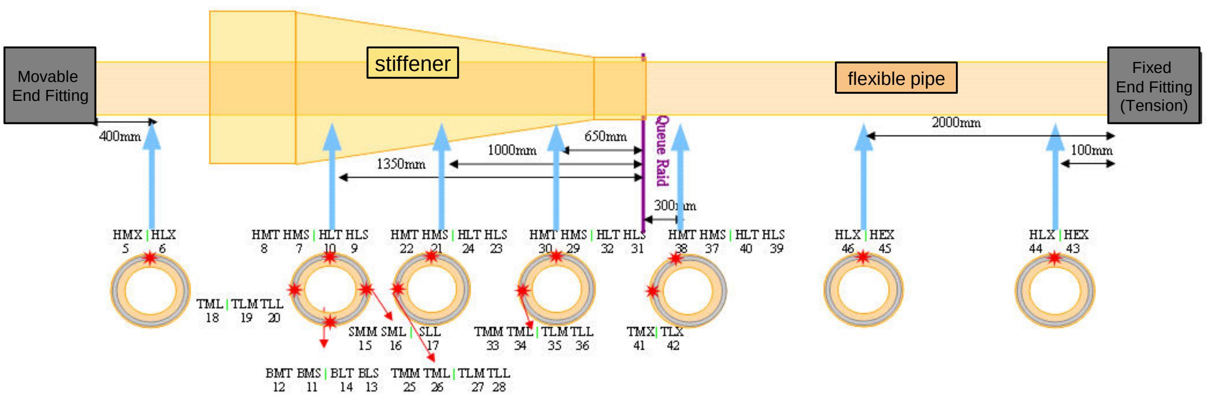

Figure 9. Strain gages locations

Numerous loading cases (combination of different internal pressures, tensions and bendings) were performed on this pipe. The comparison with numerical results, as presented in this paper, is limited to a given experimental block defined by an internal pressure of 16 MPa, a tension of 1400 kN, and cyclic rotations of the swinging table from 0° to +15°, or 0° to -15°.

3. Abaqus model description

This model describes a finite length pipe with its end fittings. It was initially developed as an expert tool, used to understand the end fitting effects, and to quantify the outer structure impact on the fatigue behaviour of the flexible pipes. It can take in account non uniform curvature distribution along the pipe. It also feeds back the engineering models in particular configurations to ensure the design safety. The model length is linked to the configuration we consider, but currently ranges up to 10 meters, limited by both CPU time and RAM required to complete the analysis.

3.1 Part, geometries and materials

As the analysis is focussed on the armour wires behaviour, the other layers are simplified. The flexible pipe finite element model presented in Figure 10 is composed of 5 layers :

- an internal kernel, modeled with shell elements, represents internal sheaths. Carcass, pressure vault and spirals, whose laying angles are closed to 90°, are represented by circumferential beams embedded in this kernel. The equivalent properties of these beams are chosen to have the same areas and inertia per unit length as these layers.
- a first armour layer,
- an anti-wear tape, meshed with shell elements,
- a second armour layer,
- an external kernel representing tapes and sheaths, meshed with shell elements.

The length of the modelled pipe was reduced to 7 meters to limit the computation times. The bending stiffener presented in the Figure 11 is composed of two parts : a rigid part representing the clamping plate, and a deformable part representing the polymer part of the stiffener. The materials properties used in the analysis are summarized in the Table 1.

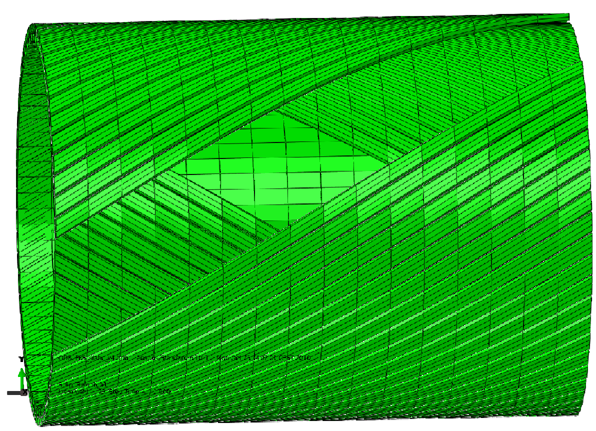

Figure 10. Flexible pipe geometry and mesh (External sheaths, anti-wear tape, and some armour wires were removed for this illustration)

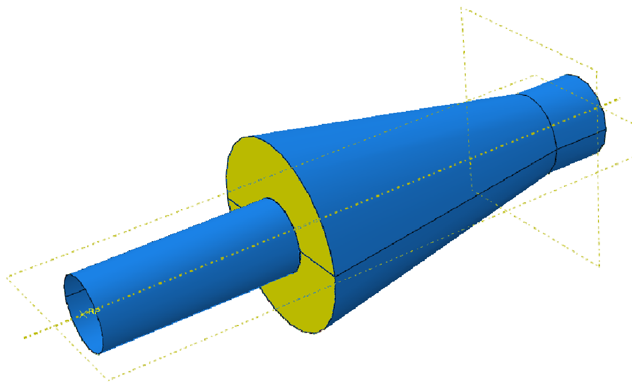

Figure 11. Geometry of the bending stiffener

Part	Material type	Parameters
Kernel	Elastic	E_{sheath}= 350 MPa , ν = 0.47 + beam properties
Armours	Elastic	E = 200 000 MPa, ν = 0.3
Anti wear tape	Elastic	E = 350 MPa, ν = 0.47
External sheath	Elastic	E = 220 MPa, ν = 0.47
Stiffener	Hyper elastic Inokuchi	user subroutine USDFLD / VSDFLD

Table 1. Summary of the material properties

3.2 Analysis type

Historically, the model was run with an explicit integration scheme solved by Abaqus/Explicit running on the IFP parallel platform on 32 CPU cores because of the large number of DOFs involved in the analysis (up to $5x10^6$). It however supposes the use of a dynamic simulation whereas this problem is quasi static. As a consequence, the simulation time must be chosen large enough to avoid structure dynamic effects but small enough to ensure its computational efficiency since the CPU time is proportional to the simulated time. Recently, and thanks to a supercomputer upgrade, the use of a static analysis with Abaqus/Standard was considered and was proved to be a very relevant choice. The results and performance of the two solvers are presented in the section 5.2.

The analysis is composed of several steps :

- step 1 : internal pressure (16 MPa) and tension (1400 kN) are applied
- step 2 and following steps : cyclic bendings of the pipe imposed by the rotation of the swinging table between 0 and 15° are simulated.

3.3 Contact interactions and couplings

All contacts interactions between the model components are considered, which allows in particular, adjacent armour wires contacts during cyclic bending tests. The way we define these interactions differs with the analysis type. For Abaqus/Explicit analysis, we use the general contact definition. When using the Abaqus/Standard solver, a Python script is used in Abaqus/CAE to define individually all the contact pairs. A Coulomb's friction law is used everywhere.

The end fittings of the pipe are modelled by a kinematic coupling (green lines on the Figure 12) between a reference point located at the centres of the end sections (green points on the Figure 12 called GIM-1.NC1 and GIM-1.NC2) and the different components of the end sections. The link between the pipe and the test rig is defined by MPC Beams (red lines on the Figure 12) between the reference points of the pipe end sections and two reference points defining the test rig located at the rotation axis of the rig (RP-Rot-Axis) and on the rotule near the traction jack (called RP-Jack). The stiffener is rigidly connected to the test rig RP_Rot-Axis reference point via a MPC Beam and the end section of the deformable part of the stiffener is linked to the pipe end section reference point GIM-1.NC1 by a kinematic coupling (blue line on the Figure 12).

3.4 Loads and boundary conditions

The loads and boundary conditions are summarized in the Figure 12. The internal pressure is applied on the internal kernel of the flexible pipe.

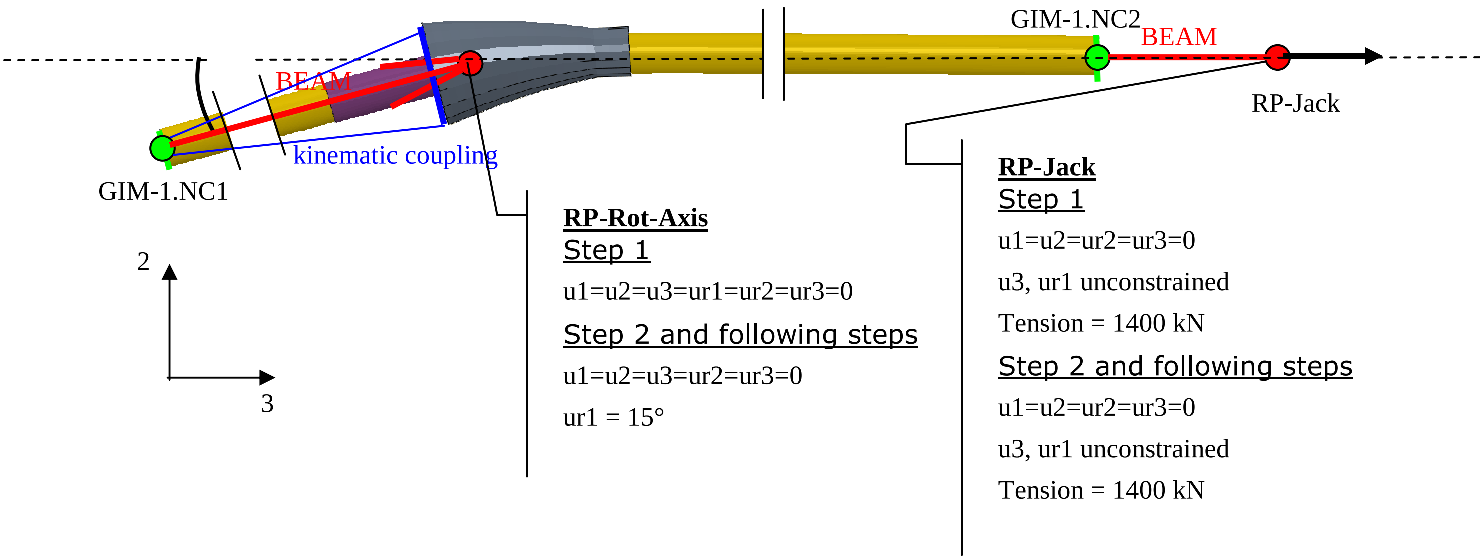

Figure 12. Couplings, loads and boundary conditions

4. Simulation results and comparison with physical tests

In this section, we compare the simulation results computed with the implicit solver Abaqus/Standard with the strain gages measurements. In the angle convention we take hereafter, 0° corresponds to the pipe extrados and 180° to the pipe intrados. Figure 13 and Figure 14 show the measured and simulated axial and transverse stresses in the armours. Axial stresses correspond to the stresses in the tangential direction of the helix, and transverse stresses are associated to the bending moment around the normal of the helix. The redundancy of the measured data (up to 4 measured curves) highlights the experimental data quality and is due to both multiple strain gages located almost at the same place on adjacent wires, and to the joint exploitation of 2 tests (0/+15° and 0/-15°).

The correlation between measured and Abaqus simulated results is very satisfying all along the pipe.

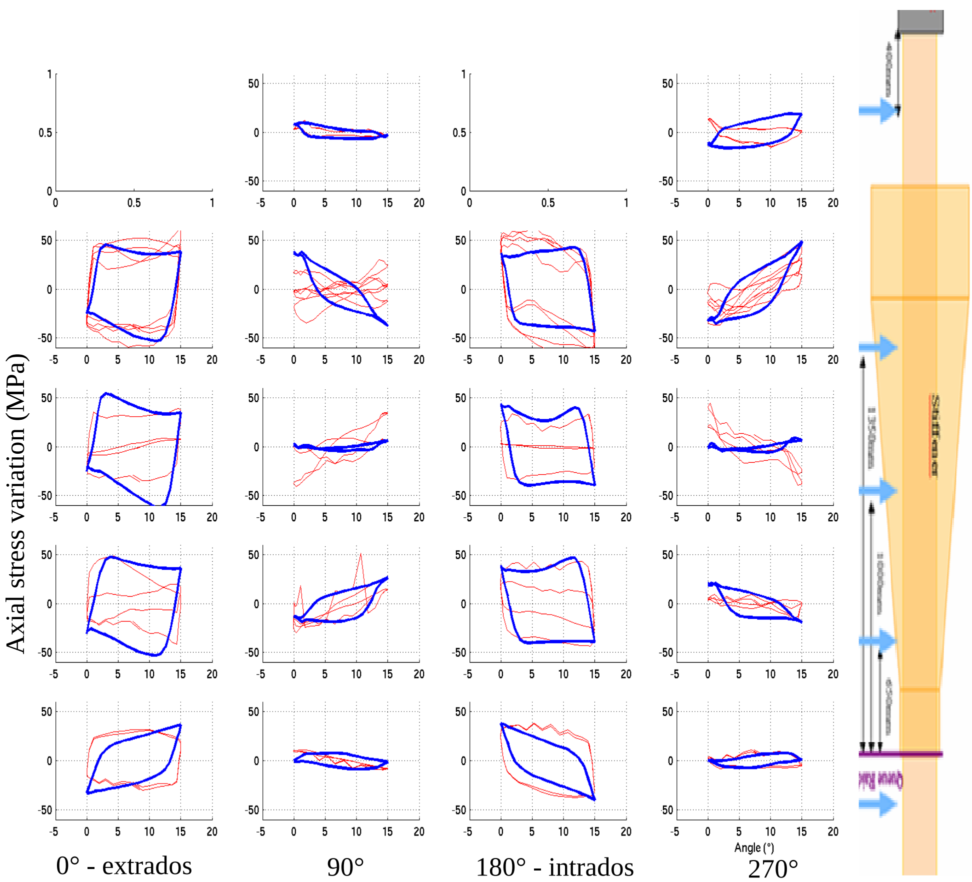

Figure 13. Measured (red) and simulated (blue) axial stress variations

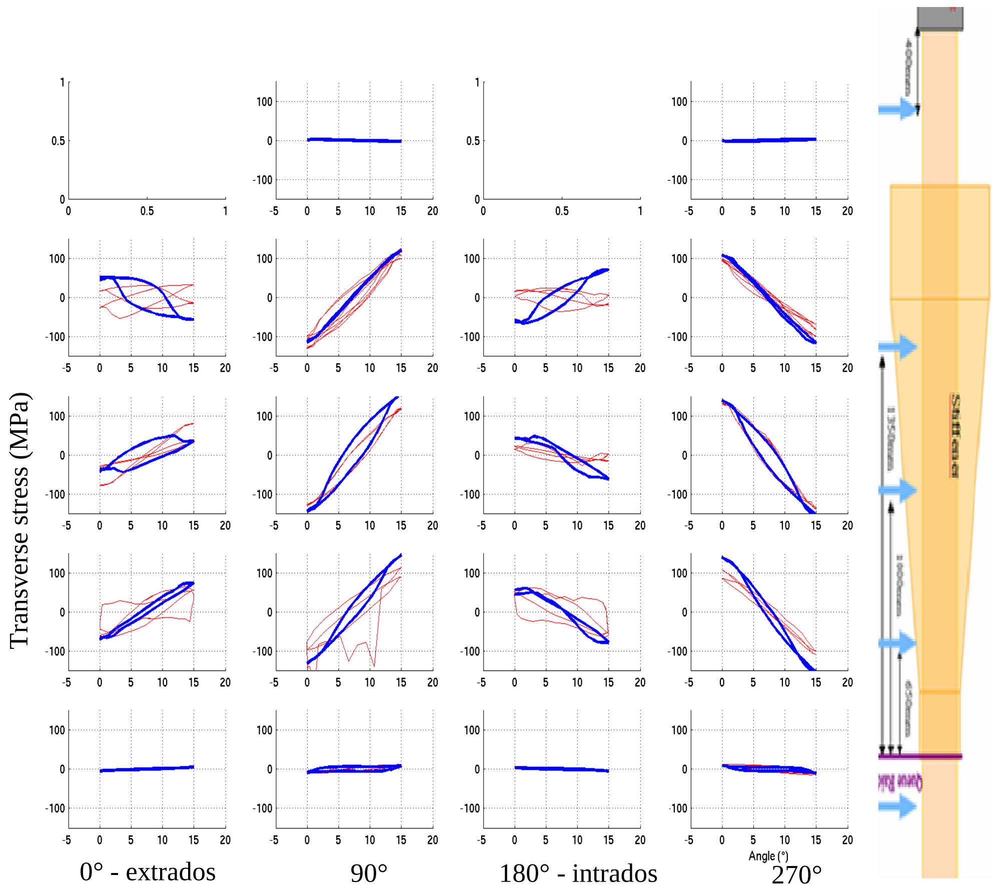

Figure 14. Measured (red) and simulated (blue) transverse stress variations

5. Results and performance comparisons of the Abaqus/Standard and Abaqus/Explicit models

5.1 Results comparison

Figure 15 presents Abaqus/Explicit and Abaqus/Standard axial stress results along each wire of the external layer. These results strongly differ near the end fitting , and Abaqus/Explicit results exhibit larger axial stresses than the physical test data in this region.

These differences were investigated and are the consequence of a dynamic effect of the Abaqus/Explicit simulation due to a too short simulated time. However, it was not possible to extend the simulated time of the explicit analysis because half a bending cycle already required almost 48h on 32 CPU cores (Table 2).

This dynamic effect is not obvious at first sight since it only affects the armour wires sliding and is not clearly identifiable in the energy outputs.

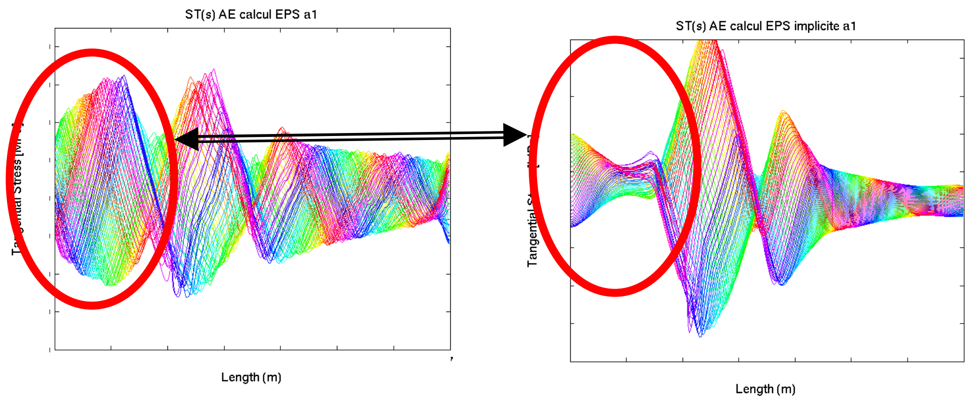

Figure 15. Abaqus/Explicit (left) and Abaqus/Standard (right) simulated axial stresses for each wire along the pipe

5.2 Performance comparison of Abaqus/Standard and Abaqus/Explicit solvers

The Table 2 presents the elapsed time necessary to complete the analysis on the two first steps i.e. the internal pressure plus tension step and the first bending, corresponding to the rotation of the swinging table from 0 to 15°. The model is composed of 1.5×10^6 elements, 2.7×10^6 nodes, with more than 5×10^6 total variables. The calculations were performed on the IFPEN cluster composed of 114 nodes including 4 CPUs AMD Barcelona@2.3 GHz (with 4 cores per CPU) and 32 GB per node. The nodes are linked through an Infiniband interconnect.

Although a too short simulated time (see section 5.1) in the explicit analysis was used, the implicit analysis is always faster with the same number of CPU cores. The speed up factor still increases when 64 CPU cores are used, and the excellent parallel performances of the standard algorithm have to be highlighted.

However, the presented calculation times are much higher by several order of magnitude (minutes vs. hours) than the engineering tool used in Technip's group : Life6 software, presented in (Leroy, 2010). The Abaqus/Standard model is then an expert tool, used to analyse specific configurations.

	Abaqus/Explicit	Abaqus/Standard	
Number of CPU cores	32	32	64
Internal pressure step + half bending cycle	48h40	32h44	19h30
Speed Up Factor (without preprocessing) * *reference : Explicit simulation	1	1.67	3

Table 2. Elapsed time associated to implicit and explicit analysis

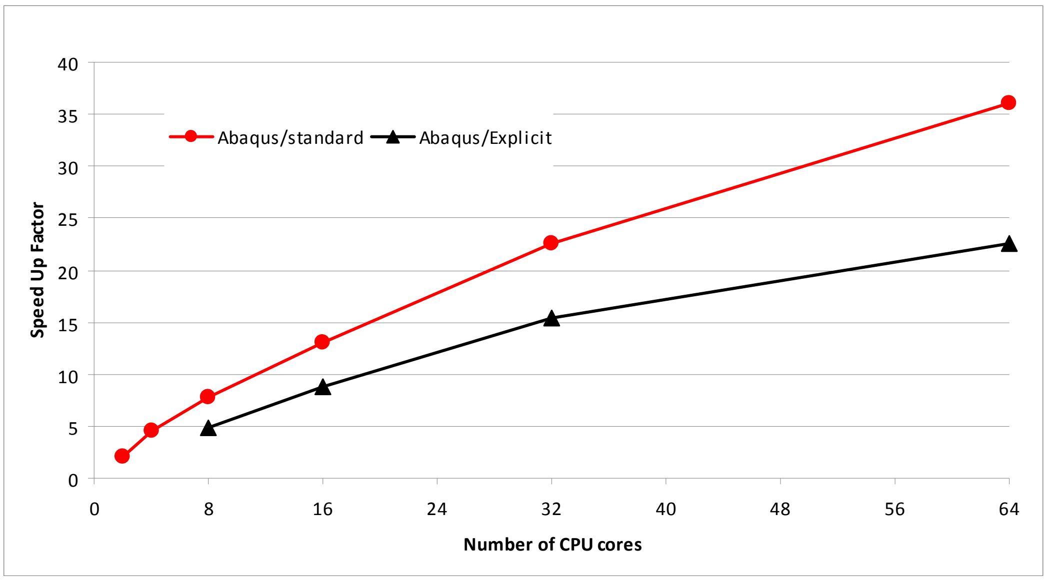

Figure 16. Speed Up Factors* for Abaqus/Standard and Abaqus/Explicit analysis. (* reference : simulation on 1 CPU core).

6. Conclusions

A cyclic bending test of a flexible pipe with internal pressure and tension was simulated by Abaqus, with both implicit static and explicit dynamic analysis. This latter requires the touchy choice of the simulated time, which must be large enough to avoid dynamic effects, but small enough to ensure the computational efficiency. For this particular test configuration, this compromise was a posteriori not reachable : despite large simulated time, dynamic effects that modified the armour stresses results occurred. These dynamic effects were not clearly identifiable from the energy outputs and may have led to misleading interpretations. The results from the static analysis performed with Abaqus/Standard were successfully compared to the experimental test results. Besides, the computational performance of Abaqus/Standard parallel solver is very satisfying with very good speed up factors up to 64 CPU cores. Some tests have still to be performed on 128 CPU cores.

However, the Abaqus/Standard do not aim at replacing Technip's design model Life6 (Leroy, 2010), but is used as a complementary expert tool to analyse specific configurations and to ensure a more efficient and reliable design.

7. References

Leroy, J-M, Perdrizet, T, Le Corre, V, and Estrier, P, “Stress Assessment in Armour Layers of Flexible Risers,” Proceedings of the ASME 2010 29th International Conference on Ocean, Offshore and Arctic Engineering, OMAE 2010, China, 2010.