Transactions, **SMiRT-24**
BEXCO, Busan, Korea - August 20-25, 2017
Division V

IMPACT ASSESSMENT METHODOLOGY FOR REINFORCED CONCRETE STRUCTURES – 3RD PHASE OF THE OCDE-IRIS BENCHMARK

J. Ezeberry[1], D. Combescure[2], J. Ayneto[3], and J. Gamarra[4]

[1] Technical Lead, Nuclear Services, Idom, Spain
[2] Technical Officer, Fusion for Energy (F4e) – European Commission, Barcelona, Spain
[3] Technical Support Staff-Analysis and Codes, Fusion for Energy (F4e) – European Commission, Barcelona, Spain
[4] Structural Engineer, Nuclear Services, Idom, Spain

ABSTRACT

The calibration of numerical tools is very important in order to validate the current assessments carried out in the ITER project. In this context, this paper is dedicated to summarize the main aspects of the methodology followed by the team F4E/IDOM in the 3rd phase of the benchmark OCDE-IRIS. This benchmark is dedicated to the improvement of the assessment of structures impacted by missiles. In particular, the third part of this program is dedicated to the propagation of the induced vibrations on a Reinforced Concrete structure impacted by a missile and the transmission of these vibrations along the connected walls and floors. For this purpose, reliable experimental information obtained from RC *mockups* is contrasted with numerical results obtained with ABAQUS software. The proposed methodology is divided in two main steps: First, following the methodology developed in previous phases of the benchmark, the impact force and the extent of damage is determined at the end of the impact from complex non-linear analysis. Second, based on the results obtained in the first step, elastic analysis with effective stiffness are carried out to calculate the vibrations of the structure. These models, properly calibrated with the experimental data, allows proper simulations showing the applicability of this methodology in the professional practice.

INTRODUCTION

The application of advanced numerical tools to complex problems within a nuclear safety environment requires some sort of validation. Precisely, this is the purpose of the IRIS benchmarks organized within the framework of the OECD/NEA (2010 – 2016). F4E is actively participating in these benchmarks. A first phase took place in 2010 and included a blind prediction exercise on two impact tests: a flexural mode test and a punching mode test. In this phase, more than 20 organizations from OECD countries presented their results in a workshop held in Paris (13-15 December 2010). In 2012 the organizers launched a second phase (IRIS-2012), to give the opportunity for updating and improving the simulations with the knowledge of the test results and with the experience gained from the computations made in 2010. The workshop was celebrated in Ottawa (17-19 October 2012). During these both phases, the team IDOM-F4E developed the methodology to assess the behaviour of impacted RC panels. This methodology was successfully applied in different non-linear assessment related to ITER project (F4E-2008-OPE-11, F4E-OMF-503-02-01).

In 2016 the OECD/NEA launched the IRIS benchmark phase 3, dedicated to the transmission of the induced vibrations from the impacted wall of a reinforced concrete structure to floors and walls which are outside the impacted area. This phase starts from the conclusions obtained in previous work and extend the validation to other regions of the structure.

CONCLUSIONS OBTAINED IN PREVIOUS PHASES

The preliminary phases of the IRIS-Benchmark (2010, 2012) laid the basis for the definition of a methodology of analysis of RC structures subjected to impacts from missiles. The conclusions and lessons obtained in previous phases are the starting point of the third phase, thus it is important to review the previous phases. The general philosophy of F4E-IDOM team was based in two points:

- The use of tools available within a regular engineering company environment (i.e. not within an R&D organization or a software developer): commercial software & engineering knowledge.
- Push the tools to their limit and try to understand where the limitations are.

ABAQUS software was used for analysis. During IRIS-2010 the performance of the missiles was predicted relatively well but the numerical test only predicts qualitatively the flexural mode test and totally failed to predict penetration of the missile in the punching mode test. The reason of the problem was found in the constitutive model selected for concrete. The use of Drucker-Prager model produced unrealistic high compression levels. Therefore, in IRIS-2012 it was decided to change to the Concrete Damaged Plasticity model of ABAQUS. This model is a modification of the Drucker-Prager model to accommodate different hardening-softening in tension and compression (Lubliner, 1989 and Lee, 1998). In this model the evolution of the yield surface is controlled by two hardening variables (one in tension + one in compression). In this phase, the material parameters of concrete were obtained from cylinder test results supplied by the organization. The same parameters are used for all simulations. The main lessons of the second phase were:

- Calibration using just one test is not enough for general purpose applications.
- Never assume that “detailed” simulation is much better than “approximate” or empirical formulae
- For general purpose applications, concrete constitutive representation needs independent state variables for tension and compression failure.
- The move from the Drucker-Prager model to the Concrete Damaged Plasticity model and the calibration with cylinder tests substantially improved the correspondence with test results.

OBJECTIVES AND SCOPE OF THIRD PHASE

The objective of the third phase of the IRIS benchmark is to analyse the vibrations induced by the impact in the structure. In order to achieve this goal, the proposed tasks consist of two Phases:

- **Phase A1:** First, an impact has been performed against a structure in the laboratory, which is named as VTT-IMPACT V1 test case. The aim of this phase is to model this structure and perform a simulation (FEM) to match the results of the experiment. In order to match the results of the finite element analysis it is necessary to change several properties of the materials, the boundaries and the global configuration.
- **Phase A2:** After deciding the final properties of the materials and their damping, a new phase is carried out. This consist of the simulation of a new model, the IRIS3 *mockup*, with the material properties used in the previous phase. The calculations are blind, so no comparison of the results can be done.

INPUT DATA

The input data provided by the organization for **Phase A1** consist of experimental results obtained in the VTT *mockup* which is a box shaped reinforced concrete structure, whose dimensions are

2000×2000×2000 mm (width×longitude×height). **Figure 1** show the experimental setup and the dimensions of the VTT *mockup*. As can be seen the boundary conditions are implemented by a system of profiles and cables to constraint the *mockup* properly. Cylinders of S355 carbon steel are used to materialise the rollers and four elastomeric bearing pads are located in each corner of the structure. The reinforced concrete has a compressive strength that varies from 50.9 MPa at walls to 59.9 MPa at floor slab. The reinforcement has a yield strength that varies from 536 MPa (Ø12 rebars) to 624.7 MPa (Ø6 rebars). The frontal wall has a width of 150 mm and the reinforcement is of rebar's Ø6 each 50 mm in horizontal and vertical directions. The triangular shear walls have a width of 150 mm and the rear wall a width of 250 mm, both are reinforced with rebar's Ø10 each 50 mm in horizontal and vertical directions. Finally, the floor slab has a width of 150 mm reinforced with Ø10 each 50 mm in the direction of the impact and with Ø6 each 50 mm in the transversal direction.

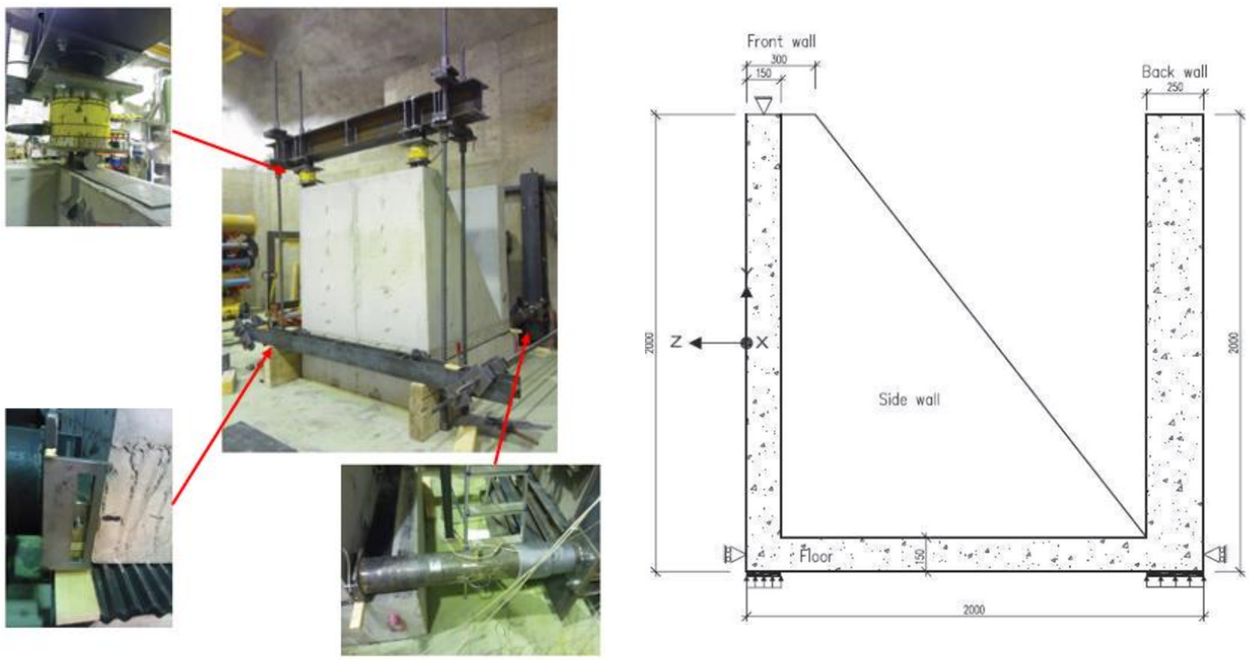

Figure 1: VTT *mockup*: experimental set-up and drawings (OECD/NEA, 2016)

Figure 2 show the shape of the impactor and the state after the impact. The missile is a stainless steel cylinder of diameter Ø254 mm, length 2200 mm and thickness 2 mm. In the front of the missile there is an end cap of the same material, length 86 mm and thickness 3 mm. In the rear part of the missile, a carbon steel plate is added to calibrate the mass. The total mass is 50.1 kg and the velocity is 113.65 m/s. In the same figure can be seen the shortening experimented during the impact which varies between 1021 and 1201 mm. The estimated impact duration is of 19 ms.

The input data provided for the phase A2 only consists of the geometry of the IRIS3 *mockup*, the materials and the size and velocities of the impactors. The reinforced concrete specimen, whose dimensions are 1730×2100×3000 mm (width×longitude×height), is presented in **Figure 3**. A frontal wall rigidly connected to two horizontal slabs at its base and a height of 2000 mm is impacted by a missile. A rear wall, connected to the same slabs complete the structure. The existing boundaries in the structure are four steel supports grade S355 rigidly connected at its base in the corners of the bottom slab. The mean concrete strength is 50 MPa and the yield strength of rebars is 500 MPa.

There are two impactors defined for the IRIS *mockup* with the shape defined in **Figure 2**. In this case, only an estimation of the maximum force produced by the impact is provided. **Missile 1** impacts a velocity of 90 m/s and has a cylinder length of 1500 mm, the total mass is 50.10 kg and the maximum force is around 0.40 to 0.50 MN. Missile 2 impacts a velocity of 170 m/s and has a cylinder length of 2400 mm, the total mass is 50.26 kg and the maximum force is around 0.90 MN.

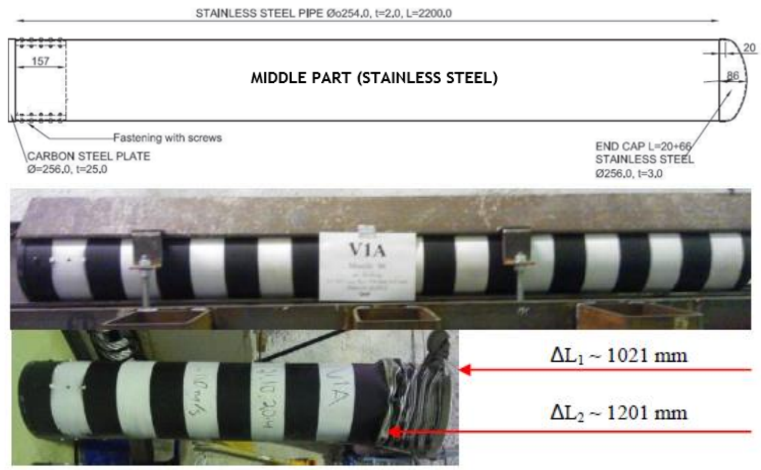

Figure 2: Missile: scheme of the missile (OECD/NEA, 2016)

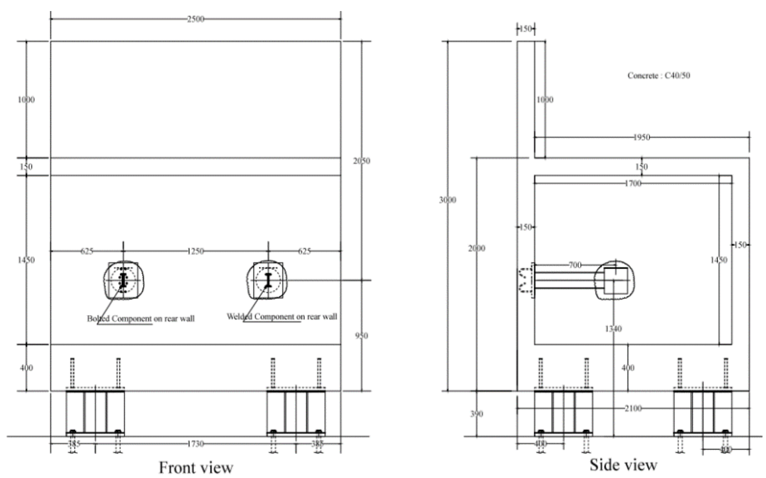

Figure 3: IRIS-3 *mockup*: drawings (OECD/NEA, 2016)

METHODOLOGY

Introduction

As concluded in previous phases of the benchmark (OECD/NEA, 2010 and 2012), the natural tool for the analysis of the impact is an explicit time-integration scheme. The main reasons of the use of ABAQUS/Explicit are:

- The impact is a high-speed dynamic event, being very costly to analyse by using an Implicit Integration Scheme. Since the load is applied very quickly and is very severe, the response of the structure changes quickly. Accurate tracking of stress waves through the structure is important for capturing the dynamic response. Since the stress waves are associated with the highest frequencies of the system, obtaining an accurate solution requires many small time increments.

- In the impact between missile and mock-up, it is generated a complex contact problem. This conditions are formulated more easily using an explicit dynamics method than using an implicit method. The result is that ABAQUS/Explicit can readily analyse problems involving complex contact interaction between many independent bodies.
- Sometimes it is important to consider damage in concrete. In this case, it is important because the degradation affects to the response against the impact. An Explicit Time Integration Scheme must be used because material degradation and failure often lead to severe convergence difficulties in implicit analysis.

Once the impact force finish, and the damage produced by the impact is maximum, the free vibration of the structure is controlled mainly by the inertial and damping forces. Without damping, unrealistic effects arise as was detected during the calibration of the model. In these cases, Implicit scheme is more effective to introduce damping. Thus, the analysis can be divided in two parts, according the time integration scheme needed: duration of impact (forced vibration, Explicit code) and free vibration (Implicit code).

Although the trespassing of results from ABAQUS\Explicit to ABAQUS\Standard it is possible, from practical point of view the problem was unaffordable. After the impact, there is a period in which the damage still evolves in the structure and convergence problems arises if Implicit code is used. On the other hand, extending the period of Explicit analysis until damage stabilizes by introducing damping implies an excessive consumption of resources avoiding this via. Thus, an engineering solution is proposed based in two different steps.

- **First**, maintaining the conclusions obtained in the previous benchmark phases, a short period of time in which the impact occurs is analysed using ABAQUS\Explicit. In this step, all the non-linearities are involved in the calculations, even an explicit representation of the missile.

- **Second**, an elastic model with effective stiffness and Implicit dynamic scheme is proposed for dynamic analysis using ABAQUS\Standard. The dynamic force is taken the same obtained in step 1. To determine the effective stiffness, from the analysis of the damaged configuration of the *mockup* after the impact, a reasonable reduction of stiffness is proposed. Based on moment-curvature relationship and engineering judgment, the value of the Young modulus is reduced. To simplify the process, only three different degraded zones are proposed in which the modulus E is considered constant: ULS zones, cracked zones and elastic zones.

As commented, the reduction of stiffness requires of engineering judgment. The way of introducing this reduction is by changing the value of E according the flexural secant stiffness (EI) of moment-curvature diagrams. To obtain these diagrams simplified RC cross-section of 1 m width are analysed. Due to the fact that the reinforcement is symmetric, the reduction of the modulus E can be considered valid for both directions. Some sensitivity analyses were carried out. It was found that the selection of an intermediate value of EI between the elastic zone (State I) and the fully cracked zone (State II) can be used for the cracked zones, and an intermediate value of EI between the fully cracked zone (State II) and the ULS strength can be used for the ULS zones.

The value of damping was approximately obtained from the dynamic signals by using the half-bandwidth method. The value obtained is closed to 5%, thus this value is adopted for the calculations of the VTT and the IRIS3 ‘blind’ simulations.

Impact analysis

Constitutive model for missile

As showed in **Figure 2**, the shortening of the missile is an input data. For the calibration of the missile a modification of the Cowper-Symonds formula is proposed for stainless steel by introducing a factor *k* varying between 0 and 1,

$$\sigma = \sigma_0 \left(1 + k \left(\frac{\dot{\varepsilon}}{100} \right)^{\frac{1}{10}} \right) \tag{1}$$

Thus, the missile is impacted against a rigid surface and the obtained shortening is contrasted against the experimental result. **Figure 4** show the difference between the real and computed shortening for different values of *k*. The best approach is obtained for *k* equal 14%.

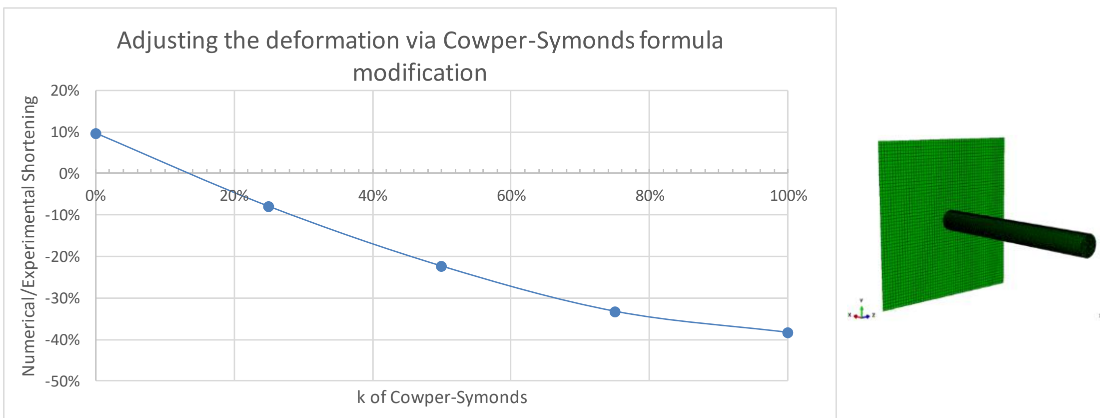

Figure 4: Calibration of the missile constitutive equation

Constitutive model for concrete

For concrete, in the previous phases (OECD/NEA, 2010 and 2012), the use of the ABAQUS concrete damaged plasticity (CDP) constitutive model was found that reproduces reasonably well the flexural tests, for cases in which the missile does not penetrate into the structure. This model uses concepts of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behaviour of concrete. It assumes that the main two failure mechanism are tensile cracking and compressive crushing of the concrete material. The evolution of the yield (or failure) surface is controlled by two hardening variables linked to failure mechanism under tension and compression loading. A slight change has been made respect the previous phases of IRIS, introducing the 1D behaviour according the proposal of Aslani (2012). This assumption implies that damage is assumed similar than for seismic cases. This change does not introduce greater uncertainties than those that exist. Other differences can be found in the definition of the parameters which defines the flow potential eccentricity which in this report is left by default equal 0.10 and the ratio of the second stress invariant on the tensile meridian to that of the compressive one which is left by default equal 2/3.

Constitutive model for reinforcement

Regarding the reinforcement steel, the same constitutive model used in previous phases is used, the von-Mises plasticity with isotropic hardening. The material parameters provided in the input data are used to define the constitutive equations for low strain rate. On the other hand, strain rate effects are computed according to CEB-187 (1988).

FE models

The missile is modelled with reduced integration shell elements (S4R). The average size mesh in the missile is 6 mm and is compatible with the failure in a concertina mode (OECD/NEA, 2010). The master surface in the contact is the stiffest part, the mock-up, whereas the external surface of the missile is the slave surface. Both parts have a similar mesh size. The number of elements is 52548 elements for the missile 2.313 m long.

A 3D solid FE of the *mockup* is carried out in ABAQUS. Reduced integration solid elements (C3D8R) are used to model the reinforced concrete. In general, a uniform mesh is used, being the mean size of the mesh equals 18.75 mm. In the impacted zone the mesh is refined by an adaptive meshing technique, reducing the size around 10 mm in order to have more accuracy (in this zone higher stress gradients occur). The number of solid elements is 486580 for VTT and 794655 for IRIS3. The reinforcement is represented by truss elements (T3D2) embedded in the concrete material, so degrees of freedom of the reinforcement nodes are eliminated and related to concrete nodes. Perfect bounding is accepted between concrete and steel. The number of elements is 40529 for VTT and 202688 for IRIS3. **Figure 5** show the models of the VTT and IRIS-3 *mockups*.

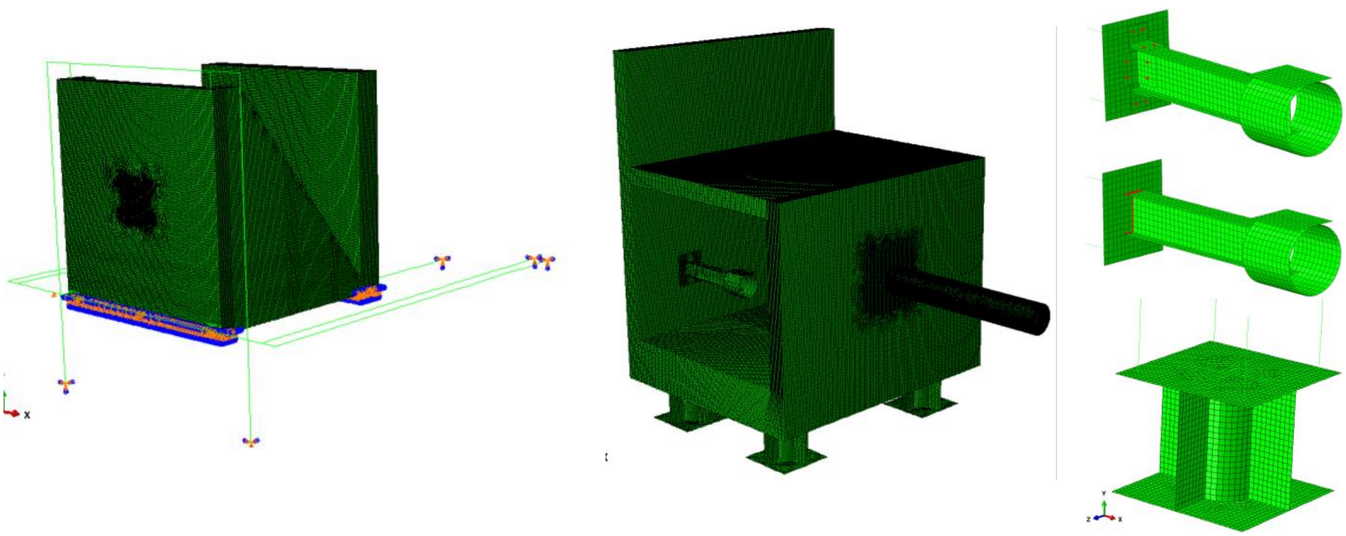

Figure 5: FE models for VTT (left) and IRIS-3 (right) *mockup*.

Boundary conditions for VTT *mockup* provided by steel framed structure is modelled by using beam elements (B31). The elastomeric bearings are orthotropic and have been modelled as springs (Springa) with different stiffness in the three directions of space. The springs are fixed to the ground in the three translation degrees of freedom. The stiffness constants of the springs have been computed to represent the same stiffness as in the elastomeric bearings. The cylinders, which constraint the structure, are modelled with beam elements (B31). In those zones in which the cylinders are in contact with the *mockup*, the rollers have been modelled as beams in contact with a contact surface on the concrete surface to restrict the movements in case the structure advance over the roller. Boundary conditions for IRIS-3 *mockup* requires that the steel supports be implemented. Reduced integration shell elements (S4R) are used to model the supports. The connection between the supports and the *mockup* is made by anchors, which are embedded in the concrete. The base of the supports is restrained.

IRIS-3 *mockup* has two cantilevered profiles connected to embedded steel plates into the rear wall by different connection methods: bolts and welds. Reduced integration shell elements are used (S4R) to model the cantilevers. The bolted profile is anchored by a system of L profiles and bolts. The profile is connected to the L90x60x80s Profiles with the bolts. The connection is made by connectors (CONN3D2) in the places where the bolts are set. L profiles are connected to the plate that is anchored to the wall of the *mockup*. The welded profile is connected in the perimeter in contact with the plate by connectors as well.

Results obtained from impact analysis

The non-linear calculations carried out during the impact allows to determine the extension of damage produced by the impact and the impact force time history. Both results are used to determine the initial conditions that produces the free vibration of the structure. **Figure 6** show the resultant impact force at the different *mockups* in spectral and time-history form. The FFT of the signals analysed indicate that the 75% of the energy is below 100 Hz and the 85% below 200 Hz. To determine a realistic maximum impact force, it was necessary to filter the signal at 500 Hz, for VVT *mockup* the maximum force is 420 kN, whereas for IRIS3 *mockups* the maximum force is equal to 410 kN (v = 90 m/s) and 820 kN (v = 170 m/s).

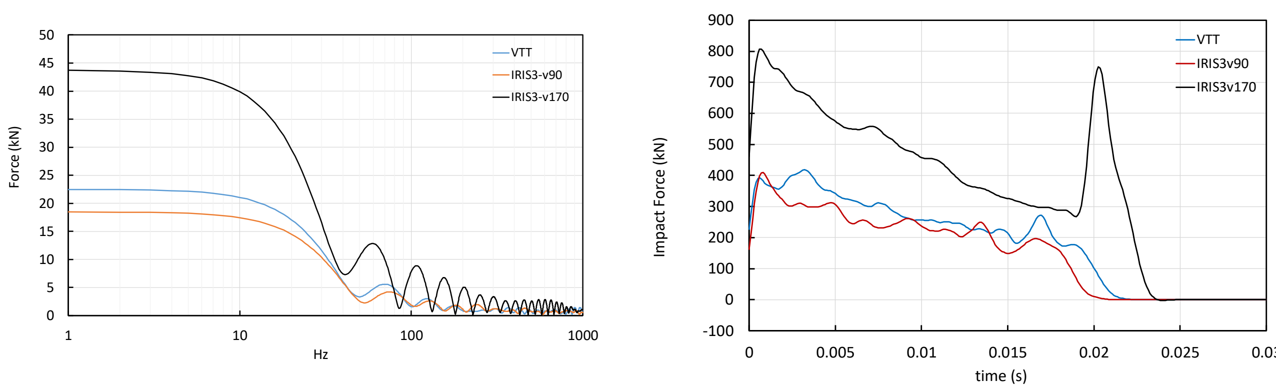

Figure 6: Impact analysis results: FFT (left) and time history (right) of the impact force (filter at 500 Hz)

Figure 7 show the extension of damage for the frontal wall of the VTT *mockup* in terms of the effective plastic strains in tension (PEEQT) and compression (PEEQ). Zones in which the value of PEEQ is close to 3.5‰ are considered in the ULS (If the PEEQ variable is analysed in steel rebars, values close to 15% can be used to determine the ULS). On the other hand, zones in which PEEQT is greater than 0.6‰ can be considered cracked. Thus, in this way, three zones can be delimited in the *mockups*: ULS, cracked and not cracked zones. The results obtained in terms of PEEQ variable reveals that the ULS zone is concentrated around the impacted zone, whereas the results obtained in terms of PEEQT variable show that the frontal wall is complete cracked after the impact. In the same way, the analysis of the rest of walls and slabs has demonstrated that the triangular transversal walls, the bottom slab and the bottom part of the rear wall is cracked, remaining the upper part of the rear wall non-cracked. The analysis of the IRIS3 *mockup* impacts (90 m/s and 170 m/s) showed a similar behaviour for the impacted panel, ULS in the zone of the impact and generalized cracking in the rest of the wall. The rest of the structure present cracked zone in coincidence with the connection between the front and rear walls and the bottom slab, and in the connection between the rear wall and the upper slab. The upper slab present some zones cracked, being more extended the damage for the impact at 170 m/s.

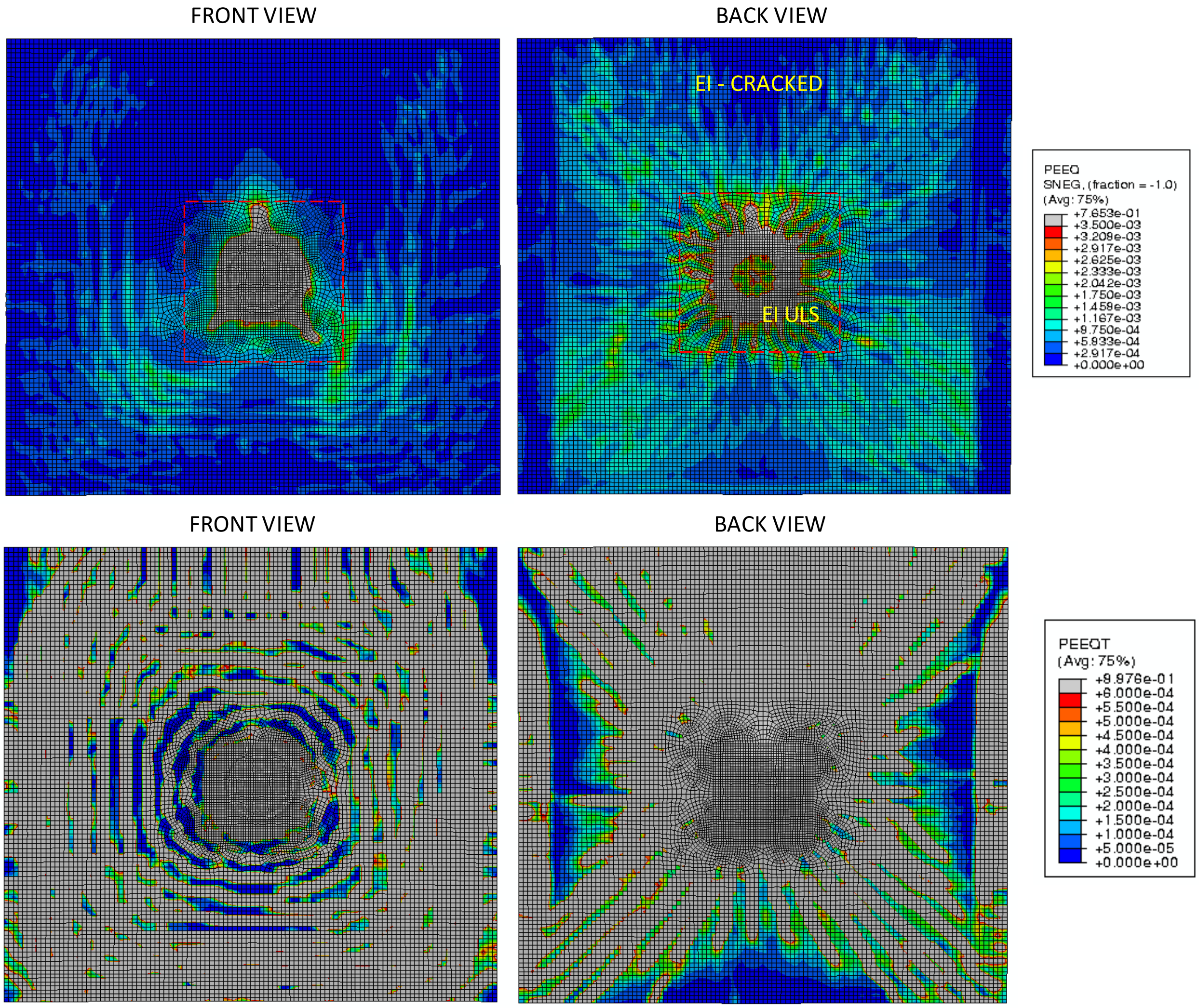

Figure 7: Extension of damage on the VTT front wall after the impact. Damage in compression (PEEQ) in the upper plot and damage produced by tensile plastic strains (PEEQT)

Free-vibration analysis

Once the extension of damage is determined, a linear elastic FE model with three different zones are defined for which the elastic modulus is adjusted according the moment-curvature relationship of the wall/slab cross section. In order to determine these diagrams, a one-meter width beam is identified for each direction. Due to the fact that the reinforcement is symmetric, the resulting diagram can be considered representative for all directions. For VTT and IRIS *mockups* was found that in the impacted zone, the materials are near the ULS, and the reduction factor for the elastic modulus is around 10-15%, in the cracked one the reduction factor is around 50-70%. Finally, elastic zones are modelled without reduction of the elastic modulus. Implicit dynamic calculations are carried out with a modal base with frequency content until 1000 Hz. The use of modal base is the most-effective way to introduce the desired damping. The impact force determined in the impact analysis is used. **Figure 8** show the comparison between the result obtained in the VTT and the experimental result. Elastic calculations are included. It can be seen that, although the magnitude of the displacement is well captured by adjusting stiffness, the model is stiffer than reality. This behaviour is verified for the rest of experimental points.

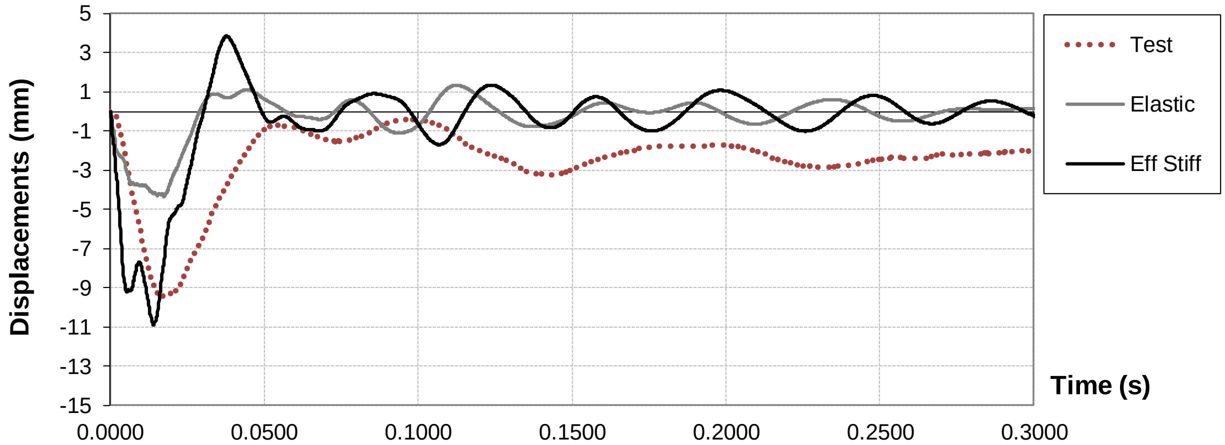

Figure 8: Comparison of displacements at the impacted zone for VTT *mockup*

CONCLUSIONS

This paper summarizes the main aspects of the methodology proposed for the study of the vibrations produced by the impact of a missile into Reinforced Concrete structures. The analysis is divided in two main steps: First, a complete non-linear analysis based on the methodology developed in previous phases of the benchmark is carried out. This analysis, based on explicit calculations, allows to determine the extent of the damage produced by the impact and the calculation of the impact force. Due to the fact that the time required by explicit calculations are strongly affected by damping, and that damping has a strong influence in the free vibrations of the structure, a second step is proposed. In this step, Implicit calculations with a linear elastic FE model with adjusted stiffness are carried out. To simplify the approach, three main zones in which the elastic modulus is fixed are proposed: ULS, cracked and non-cracked. Values taken from moment-curvature diagrams are used to estimate the reduction of stiffness. The obtained results reveals that the model is stiffer than reality, but the magnitude of the displacements are close to the measured vibration (at least in the zone of the impact). Although the methodology requires some improvements, in order to capture adequately the natural frequencies of the damaged structure, the use of modal based superposition techniques strongly reduces the required time to obtain results, making this option attractive to the analysis of induced vibrations in Nuclear and other Facilities.

REFERENCES

OECD Nuclear Energy Agency (NEA) (2010). “Improving Robustness Assessment Methodologies for Structures Impacted by Missiles, IRIS-2010.”

OECD Nuclear Energy Agency (NEA) (2012). “Improving Robustness Assessment of Structures Impacted by Missiles, IRIS-2012.”

OECD Nuclear Energy Agency (NEA) (2016). “Improving Robustness Assessment of Structures Impacted by a large missile at medium velocity, IRIS phase 3.”

F4E-2008-OPE-11 (ES-AC) (2009-2014). “Engineering Support in the Area of Civil Engineering Analysis: Seismic Analyses, Effect of explosions and impacts, Structural Analyses of ITER Buildings”

F4E-OMF-503-02-01 (2015-2019). “Engineering Support in the Area of Structural Analysis: Multiple Framework Contract for the Provision of Dynamic analysis of building and mechanical components of the ITER facility.”

Lubliner J., O. J., O. S., and O. E. (1989). “A plastic-damage model for concrete,” *Int. J. Solids Struct.*, vol. 25, no. 3, pp. 299–326.

Lee J. and F. G.L. (1998). “Plastic-damage model for cyclic loading of concrete structures,” *ASCE J. Eng. Mech.*, vol. 124, no. 8.

F. and R. J. Aslani (2012). “Stress-strain model for concrete under cyclic loading.” *Magazine of Concrete Research* (8). Research Online, pp. 673–685.

CEB (1988). “CEB-187: Concrete Structures under Impact and Impulsive Loading. Synthesis Report.”