UK EPR	Title: PCSR – Appendix 3 – Computer Codes used in Chapter 3
	UKEPR-0002-310 Issue 02

Total number of pages: 53	Page No.: I / V
Chapter Pilot: M. BERNARD *Name/Initials* *Date 12-10-2012*	
Approved for EDF by: A. MARECHAL *pp. F. BRISSE* *Name/Initials* *Date 15-10-2012*	Approved for AREVA by: G. CRAIG *Name/Initials* *Date 12-10-2012*

REVISION HISTORY

Issue	Description	Date
00	PCSR June 2009 update: First issue.	27-06-2009
01	Consolidated Step 4 PCSR update: - Minor editorial changes - §1 ASTER: Extent of code Aster used in the design of UK EPR clarified. Section updated. - §4 MAGIC: Labelling of references adjusted - §6 PRECONT: Addition of PRECONT modules software and reference documents - §7 ProMISS3D: Addition of details for the validation section (international and national benchmarks), addition of a list of functionalities. References added and adjusted - §13 reference added (code website) - New §14 SOFISTIK added - New §15 CYBERQUAKE added - New §16 ANSYS added - New §17 COBEF added - New §18 HERCULE added - New §19 NASTRAN added	30-03-2011
02	Consolidated PCSR update: - References listed under each numbered section or sub-section heading numbered [Ref-1], [Ref-2], [Ref-3], etc - Minor editorial changes - Update of references to English translations (§7, §8, §12) - Clarification of text (§13.1, §17.1, §18.1, §18.3)	15-10-2012

Copyright © 2012

AREVA NP & EDF
All Rights Reserved

This document has been prepared by or on behalf of AREVA NP and EDF SA in connection with their request for generic design assessment of the EPR™ design by the UK nuclear regulatory authorities. This document is the property of AREVA NP and EDF SA.

Although due care has been taken in compiling the content of this document, neither AREVA NP, EDF SA nor any of their respective affiliates accept any reliability in respect to any errors, omissions or inaccuracies contained or referred to in it.

All intellectual property rights in the content of this document are owned by AREVA NP, EDF SA, their respective affiliates and their respective licensors. You are permitted to download and print content from this document solely for your own internal purposes and/or personal use. The document content must not be copied or reproduced, used or otherwise dealt with for any other reason. You are not entitled to modify or redistribute the content of this document without the express written permission of AREVA NP and EDF SA. This document and any copies that have been made of it must be returned to AREVA NP or EDF SA on their request.

Trade marks, logos and brand names used in this document are owned by AREVA NP, EDF SA, their respective affiliates or other licensors. No rights are granted to use any of them without the prior written permission of the owner.

Trade Mark

EPR™ is an AREVA Trade Mark.

For information address:

AREVA NP SAS
Tour AREVA
92084 Paris La Défense Cedex
France

EDF
Division Ingénierie Nucléaire
Centre National d'Equipement Nucléaire
165-173, avenue Pierre Brossolette
BP900
92542 Montrouge
France

TABLE OF CONTENTS

APPENDIX 3 – COMPUTER CODES USED IN CHAPTER 3

1. ASTER

1.1. BRIEF DESCRIPTION OF THE CODE

Code_Aster [Ref-1] is a calculation code based on the finite element method. It offers a full range of multiphysics analysis and modelling methods that go well beyond the standard functions of a thermomechanical calculation code: from seismic analysis to porous media via acoustics, fatigue, stochastic dynamics, etc. Its modelling, algorithms and solvers are constantly undergoing development to improve their robustness and thoroughness and add to the functionality. There are currently 1,200,000 lines of code and 200 operators. With a resolutely open architecture, it can be linked, coupled and encapsulated in numerous ways, giving the user considerable freedom of choice.

This software is subject to rigorous QA procedures [Ref-2] that include independent validations, a reference of 2,000 test cases, maintenance of 13,000 pages of documentation, source code management and version qualification. The Aster quality criteria governing the development and distribution of the code are based on an auditable quality framework that meets the requirements of the EDF Nuclear Structures Safety Authority. These criteria constitute the Aster Program Quality Plan and are defined in the code Administration Manual. The theoretical foundations of Aster models are documented in the Reference Manuals. The independent validation of the operational versions of the code, which is carried out by external calculation companies, deals with the program’s conformity with respect to its documentation, the actual coverage of the declared analysis domains and non-regression in terms of precision/performance. The Quality Sheet accompanying each operating version is updated for each sub-version.

1.2. EXTENT OF CODE_ASTER USED IN THE DESIGN OF UK EPR

Code_Aster is used in the design of the civil structures of the UK EPR. The extent of use of Code_Aster and its application in the civil and structural design of the UK EPR are as follows:

- Linear elastic analysis of all the buildings on the common raft foundation
- Seismic analyses are carried out using linear elastic response spectrum analysis method.
- Floor response spectra are obtained by the linear response history analysis method in the time domain in the modal basis.
- In the seismic analysis of structures including dynamic soil-structure interaction effects, the impedance functions are calculated using a further code (MISS3D). The stiffness term of an impedance function is modelled in Code_Aster by linear spring elements, while the damping term is modelled by the modal damping method.

- The non-linear behaviour resulting from the contact and the uplifting between the structure and the ground is taken into account if necessary. Only the contact surface between the structure and the ground is non-linear (contact and uplifting behaviour), with the structure remaining linear elastic.

1.3. FULL EXTENT OF CODE_ASTER CAPABILITIES

1.3.1. Phenomena treated by ASTER

Code_Aster is capable of treating a wide range of phenomena, as outlined below:

- Mechanical (Static, quasi-static, dynamic on a physical or modal basis …)
- Thermal (Stationary, transient, linear or non-linear, fixed or moving reference coordinate system)
- Associated phenomena (Acoustics, Metallurgy, Hydration and drying)

1.3.2. Multiphysics capability

Code_Aster has an extensive multiphysics capability, achieved via internal links within the code itself as well as via coupling to external codes.

- Internal links with thermics
- Internal links with mechanics
- Internal couplings
- External coupling with other codes: Soil-(Fluid)-Structure (MISS3D)
- External links with other codes

1.3.3. Loads modelled

A number of different mechanical and thermal loads can be modelled in Code_Aster.

1.3.4. Non-linearities in static and dynamic calculations

- Geometrics
    - Geometric updating, large strains, large rotations
    - Follower forces
    - Continuation methods: in displacement, by arc length, in strain, by criterion output
    - Load discharge and non-radiality indicators

- Contact and friction: by a discrete contact method (active stresses, penalisation, conjugated projected gradient) or by an augmented Lagrangian method
- First order buckling

- Materials

Code_Aster includes 95 material behaviour laws. Effects that can be treated include the following:

  - Linear and non-linear elasticity
  - Non-linear hyperelasticity
  - Local elastoplasticity and elastoplasticity with gradient formulation
  - Non-linear viscoelasticity
  - Local and with gradient formulation damage
  - Elastoviscoplasticity
  - Metallurgical effects
  - Material data dependent on temperature, metallurgical condition, hydration, drying and fluence
  - Progressive strain
  - Hydration, shrinkage and creep of concrete
  - Geomaterials

1.3.5. Dynamics capability

Various types of dynamic analysis are available, as listed below:

- Modal analysis
- Linear transient response
- Transient response with local non-linearities (on modal basis)
- Harmonic response
- Random response
- Direct non-linear analysis
- Sub-structuring
- Seismic analysis

- Extrapolation of experimental measurements

1.3.6. Interactions

- Fluid-Structure
- Soil-Structure and Soil-Fluid-Structure
  - Absorbent boundary elements
  - Frequential coupling with **MISS$_3$D**

1.3.7. Thermal analysis capability

It is possible to perform different types of thermal analysis, including:

- Linear and non-linear thermics
- Metallurgical changes
- Thermal treatments and welding

1.3.8. Application to Geotechnical Civil Engineering

- Behaviour laws for concrete (reinforced or pre-stressed) and geomaterials
- Hydration, drying and basic creep at different time scales
- Passive reinforcement or pre-stressing effect with elastoplastic behaviour: bar, grids and membrane
- Creep-cracking coupling
- Thermohydromechanics (porous media, formulation in effective stresses, behaviour laws in kit form)
- Specific loads (hydric and gaseous flows)
- Excavation procedure

1.3.9. Fracture, damage, fatigue and collapse of structures

Code_Aster is capable of treating fracture, damage, fatigue and collapse phenomena.

1.3.10. Solvers

- Linear (LDLT, multifrontal, PCG,MUMPS, FETI)
- Non-linear (Newton, etc.)

- Integration schemes (Runge-Kutta, Newmark, adaptatives, etc.)
- Modals (Power, Lanczos, IRAM, etc.)
- Extended parameter setting. Several strategies for re-numbering, storage, pre-conditioning, post-verification

2. EUROPLEXUS

2.1. BRIEF DESCRIPTION OF THE CODE

EUROPLEXUS [Ref-1] is a general finite element code for the non-linear dynamic analysis of fluid-structure systems subjected to fast transient dynamic loading. It has been jointly developed since 1999 by CEA (“Commissariat à l’Energie Atomique et aux Energies Alternatives”, Saclay, France), EC-JRC (“European Commission – Joint Research Centre”, Ispra) and EDF R&D under a collaboration contract.

It stems from CEA's CASTEM-PLEXUS (a program belonging to the CASTEM system) and the previous CEA-EC joint product PLEXIS-3C. EUROPLEXUS is the property of CEA and EC. A commercial version distributed by SAMTECH is used for APC studies.

EUROPLEXUS is interfaced to pre- and post-processing programs of the CASTEM system (GIBI) that enable the automatic meshing of the structure and the post-processing of the results. EUROPLEXUS is also compatible with the MED format (assimilated to the Salome software environment), IDEAS pre- and post-processing as well as with the ParaView post-processing program.

2.2. MATHEMATICAL METHOD

The EUROPLEXUS is a general computer code performing 1-D, 2-D or 3-D solid (continua, shells and beams) and fluid analyses. In the present context only its capabilities for structural analyses are relevant.

The code is suitable for describing fast dynamic phenomena with the possibility of taking into account the geometrical non-linearities (i.e. large displacements, large rotations, large strains), and the non-linear material behaviour (e.g. plasticity, damage, etc).

The program uses exclusively explicit transient algorithms for the time integration. There are several spatial discretisation methods available in the code, but in this context, only those features involving the Finite Element Method are noted.

2.3. VALIDATION

In the framework of the safety assessment, EUROPLEXUS is used mainly to assess the non-linear bending behaviour of shell structures under dynamic loading. It has been validated by comparisons with experimental results obtained from reduced scale mock-ups. Validation of larger examples has been done by comparison against the numerical solutions of other codes. All these studies are subject to classification.

3. JUPITER

JUPITER [Ref-1] is a general computer code for beam model structural analyses.

It was developed by FRAMATOME in order to perform the reactor coolant loop static and dynamic linear analyses.

The eigenvalue analysis is performed by a HOUSEHOLDER-QL method [Ref-2].

The Guyan reduction method is also available [Ref-3].

Dynamic analysis is performed by means of modal and spectral analysis.

JUPITER also offers the possibility of computing the response of a structure to time dependent loadings, by use of a direct method for solving the equations of systems (β-Newmark method) in linear analysis, including taking into account stops or clearances.

4. MAGIC

4.1. BRIEF DESCRIPTION OF THE CODE

The MAGIC software [Ref-1] is a digital simulation tool developed by EDF R&D. It calculates, the spread of a fire in a building consisting of one or more rooms that communicate with each other, on the basis of data entered by the user that characterise the geometry, materials present and the heat sources of the fire.

4.2. PHYSICAL MODELLING

MAGIC is a conventional thermal model for the simulation of fire using a zonal approach, capable of handling problems involving several rooms that communicate with each other. Each room is divided up into two supposedly homogeneous zones: with smoke floating above cooler air.

The mass and energy budgets drawn up for each zone, combined with the equation for the diffusion of heat in walls discretised according to their thickness, govern the aero-thermodynamic behaviour of the building during the fire.

Specific models generated on the basis of appropriate experimentation are used to interpret the various phenomena involved:

- the pyrolysis of burning fuels
- combustion in the gas phase, which depends on the properties of the products given off and the air supply due to the entrainment of the plume
- the production of smoke and unburned matter, the properties of which depend on the fuels
- the plume of heat that forms above each source in different configurations: circular or linear fire area, at different heights above the openings
- the exchanges of heat by convection and radiation between the flame, the air, the smoke, the walls and the environment
- natural flows through the openings (vertical and horizontal). These flows link the compartments with each other and the outside (in the event of internal flow, we model fire areas produced by the ignition of unburned gases)
- natural or mechanical ventilation systems
- the stresses and thermal behaviour of sensitive elements to deduce their failure or ignition
- thermal behaviour and combustion of electric cables

MAGIC calculates concentrations of oxygen and unburned matter of the different areas of gas. Secondary fire areas may also be generated by the initial fire areas: a trigger threshold is activated with a delay or according to the temperatures attained by the combustible masses (secondary fire areas such as cables).

From a geometrical point of view, MAGIC considers a set of rectangular rooms, the edges of which are parallel to the reference axes. These rooms communicate with each other and with the outside by horizontal or vertical openings.

We therefore obtain the variations in the aeraulic and thermal conditions in the installation, on the basis of the geometrical and thermal characteristics of the rooms and their connections, for one or more fire areas. Calculated quantities include:

- the temperatures of the hot and cold areas
- the concentrations of oxygen and unburned matter
- the height at which smoke enters each compartment
- the mass flow rates of air and smoke passing through the openings and ventilation systems
- the pressures in each room at floor level
- the temperatures on the surface of the walls
- thermal flux (radiant and total) exchanged by the targets positioned by the user

Other, more specific results (plume temperatures, consumption of fuel, temperature of flows passing through the ventilation systems, temperatures of targets, etc.) are also accessible.

4.3. VALIDATION

The ability of MAGIC to predict thermal effects quantitatively during a fire in a building with one or more rooms is subject to a very complete validation file [Ref-1]. This is based on the comparison of the results obtained with the MAGIC code with full scale reference tests, for single or multiple room configurations The reference tests, approximately twenty selected from a base of more than 100 tests, have various origins (national and international) and cover a wide range of conditions, as follows:

- Volume of rooms: 11 to 3220 m^3
- Power of fire: 100 kW to 4.4 MW
- Ventilation: mechanical or natural with openings of fixed or variable dimensions
- Types of configurations: single and multiple compartments
- Types of fuels: liquid (oil, etc.) or solid (cables, etc.)
- Configuration of the fire areas: circular or linear

4.4. RECOGNITION OF THE CODE

The code has been widely compared with its competitors (Flamme_S, COMPBRN, FIVE, CFAST, FDS, etc.) during several international benchmarking exercises:

- Comparison France: Flamme_S (IRSN) in 1997
- Comparison US: COMPBRN, FIVE, CFAST in 1998
- IRSN-NRC Benchmark (CFAST, MAGIC, Flamme_S, JASMINE, FDS, ...)
- MAGIC has been checked and validated in accordance with the ASTM E1355 method as part of the V&V project (Verification & validation of Selected Fire Models For Nuclear Power Plant Applications) led by the NRC (NUREG 1824)
- OECD PRISME project (2006-2010): International programme of fire testing in rooms with multiple compartments

5. MANTA

The MANTA code, used for overpressure calculations, is described in the chapter related to accident analysis (see section 14 of Appendix 14A).

6. PRECONT – COMPUTER ASSISTED ANALYSIS AND DESIGN OF PRE-STRESSED CONCRETE CONTAINMENTS

6.1. BRIEF DESCRIPTION OF THE CODE

The PRECONT software [Ref-1] is used to design containments for PWRs. It may be considered to be a true specialised CAD system with geometrical design and structural analysis functionalities. It is used by the industrial designers to create geometrical shapes and by metal and reinforced concrete structural engineers for the detailed study of pre-stressing.

Whilst the overall sizing of a pre-stressing system is generally easy (simple equilibrium statements), the detailed study of pre-stressing is very complex:

- Geometrical complexity:
    - the cables must bypass many penetrations which may be large (equipment hatch, air lock)
    - cables must be laid in two layers in the wall to respect the geometrical constraints and minimum distance criteria
    - the junction between the cylindrical part and the dome is very complex in terms of pre-stressing
- The demands of structural analysis:
    - Any deviation or non-centred position of a cable generates considerable forces and bending moments, which locally affect the behaviour of the structure. It is therefore essential to model the geometry of the cables with the greatest precision possible

6.2. MODELLING OF CABLES

- Basic hypotheses:

In order to simplify the description of cables in three dimensions, we assume that each cable is defined by two complementary projections on a plane.

Z is the axis of symmetry of the containment

(x, y, z) is the system of Cartesian coordinates

(α, R, z) is the system of cylindrical coordinates associated with (x, y, z)

The projections used are as follows:

- the cylindrical projection (α, R0, z) on certain reference radii R0
- the vertical section (R, Z)

- the projection on a plane (x, y or α, R)

For each projection on a plane (except (x, y) for the dome cables), only arcs of circles and straight segments are considered. For the dome cables, (x, y), the arc of an ellipse is also included.

- *Data input procedure:*

Geometrical data may be entered interactively using a CAD work station (graphic tablet, menu of programmable commands, graphic display) or by direct encoding. A minimum quantity of data is necessary to define deviations, since all intermediate geometrical calculations are carried out automatically.

- *Discretisation procedure:*

Each cable is discretised into a succession of straight bars connected by joints. Accordingly, the main and secondary views are combined and successive points are calculated according to maximum and minimum curve spacing criteria, which give a true, three-dimensional model for each cable. Two or three hundred points are a common order of magnitude for a normally out-of-true cable. The geometrical data base is completed by the results of the discretisation phase. This process does not require additional data and is carried out entirely automatically.

6.3. DIAGRAMS AND ELEMENTARY CALCULATIONS

- *Diagrams:*

Many types of diagrams may be drawn using the geometrical data base, either from data projected on to a plane or a discretised 3D model:

- control diagrams are used to check the validity of the model (no crossing of cables, correct deviations around openings, etc.)
- indicative diagrams are generally drawn using data projected on to a plane, but digital data, such as the spacing of cables and the length of the by-passed parts are calculated using the 3D model
- the diagrams described may be to any scale and for any part of the containment, merely using the conventional technique of graphic windowing

The 3D model is used to plot any transverse, horizontal or vertical cross-section of the containment immediately, by showing the layout of the cables in the thickness of the wall.

- *Construction drawings:*

In addition to the as-built drawings, construction drawings may be produced using geometrical data: for example, sheets indicating the height of horizontal cables in certain angular positions have been used as a construction document for most containments built in France.

- *Calculation of losses of tension in cables:*

The 3D representation of cables allows for the simple, precise calculation of losses of tension, according to conventional formulae. Losses due to the relaxation of steel, the normal shrinkage and creep of concrete and the slippage of anchors are easily integrated in the calculations.

- *Checking tensioning:*

In order to check the tensioning phase, the planned elongation values are calculated for each cable to compare them with the values measured on site. Since any differences have a significant effect on the real elongation value, a calculation taking the exact geometry into account is necessary. This check is used to detect anomalies on site (such as the failures of pre-stressed components).

6.4. STRUCTURAL ANALYSIS: INTERFACE WITH FINITE ELEMENTS

The construction survey of the containment generally includes a large number of finite element models. In particular, certain critical zones, such as the equipment hatch, stiffeners and the connection of the cylindrical part to the dome, must be studied with great care.

- *Pre-stressing forces:*

By first considering a cable as a succession of smalls bars connected to each other, we deduce that the concrete should affect each connection by a force that balances the tensions between the bars to ensure the equilibrium of the cable. That is why it is simple to calculate the forces applied using a 3D model and calculated tensions. The pre-stressing system is then reduced to a set of elementary forces associated with points in the three-dimensional space.

- *Algorithm described:*

For models which may include several hundred 3D elements and tens of thousands of pre-stressed points, we must reduce the computer's calculation time by eliminating unnecessary calculations for points “obviously” outside the elements.

A list of potential points is drawn up for each element of the model according to a limit criterion.

For a containment sub-model, special processing is required to determine the intersections between each cable and the limits of the model and to reject points outside.

6.5. PRECONT SOFTWARE

A list of PRECONT software modules and reference documents is provided in Appendix 3A.6 – Table 1. A list of PRECONT software test cases is provided in Appendix 3A.6 – Table 2.

APPENDIX 3A.6 - TABLE 1

PRECONT Software: Modules and Reference Documents [Ref-1] to [Ref-15]

Module Name	Function description	Document Reference Number	
		Revision A	Revision B
Allong calpdom	Calculation of cable elongation during tensioning Decomposition of dome cables into straight segments and circular arcs	60590011 UM 008-(10/1997) 09456010 RP 004-(10/2008)	
Coth Cotv Cotdom	Calculation of intersection point elevations of horizontal cables with regularly spaced radial planes Calculation of vertical cable spacing at a given elevation Calculation of elevation of a dome cable at the intersection with a cylinder with a given radius	09456010 RP 001-(10/2008) 09456010 RP 002-(10/2008) 09456010 RP 003-(10/2008)	
Defcab defcom	Encoding of horizontal and vertical cables (original version replaced by EDICAB) Encoding and discretisation of dome cables (original version replaced by EDICAB)	09456010 RP 007-(10/2008) 09456010 RP 005-(10/2008)	
Destens Discr	Calculation and design of cable tension as a function of curvilinear abscissa Discretisation of an encoded horizontal or vertical cable (see EDICAB)	60590011 UM 007-(10/1997)	
Discrdom Ecartdom/distdom edicab	Discretisation of encoded dome (using DEFDOM) Calculation of spacing (on a plane) between points of intersection of a dome cable layer with cylinder of a given radius Encoding and discretisation of all types of cables with graphic aid (replaces DEFCAB, DEFDOM, DISCR and DISCRDOM)	09456010 RP 006-(10/2008) 09456010 RP 008-(10/2008) 60590011 UM 002-(10/1997)	 60590011 UM 002-(02/1998)
Fnodal mistens	Calculation of vector loading due to pre-stressing applied to finite element model Calculation of cable tension as a function of curvilinear abscissa	60590011 UM 006-(12/1997) 09456010 RP 009-(10/2008)	
Recol	Re-fusing of vertical and gamma cables	09456010 RP 010-(10/2008)	

APPENDIX 3A.6 - TABLE 2

PRECONT Software: Test Cases

Module Name	Function description	Test Case(s)
Allong	Calculation of cable elongation during tensioning	Test 1: Calculation of elongation and tension along the cable
Destens	Calculation and design of cable tension as a function of curvilinear abscissa	Test 1: visualisation of tension along a cable
Discrdom	Discretisation of dome cables	Test 1: visualisation of cables
Ecartdom/distdom	Calculation of spacing (on a plane) between intersection points of a dome cable layer with cylinder of a given radius	Test 1: visualisation of dome cable spacing
Edicab	Encoding and discretisation of all types of cables with graphic aid (replaces DEFCAB, DEFDOM, DISCR and DISCRDOM)	Test 1: visualisation of cables
		Test 1: prestressed beam on simple support Test 2: horizontal prestressing of EPR containment cylindrical zone
Fnodal	Calculation of vector loading due to prestressing force applied to finite element model	Test 3: prestressing of EPR containment central dome area

7. PROMISS3D

7.1. BRIEF DESCRIPTION OF THE CODE

MISS3D [Ref-1] is a Boundary Element Methods (BEM) analysis code dedicated to Soil Structure Interaction (SSI), Structure Soil Structure Interaction (SSSI) and Soil Fluid Structure Interaction (SFSI) under dynamic analysis.

ProMISS3D is a QA tool used as an interface between the software MISS3D and Code_Aster.

The code was developed by Ecole Centrale de Paris (ECP) and since 1996, EDF and ECP have continued to develop ProMISS3D in Code_Aster interface in order to couple both codes for SSI calculations in earthquake engineering (dams, nuclear facilities…).

ProMISS3D works with pre- and post-processing programs enabling the automatic meshing of the structure and processing of the results. It is externally coupled with Code_Aster and is developed under QA procedures including independent validations, test cases, documentation and qualification of versions [Ref-2].

7.2. METHOD

ProMISS3D is employed to solve wave propagation in fluid or elastic domains for stratified media using boundary elements. It works in the frequency domain.

The geometry and the behaviour laws are considered as linear. The software relies on the hypothesis that the domain is divided into sub-domains in order to mesh only the interfaces between the sub-domains and not the interfaces between soil layers.

7.3. VALIDATION

The validation of the ability of the software to solve wave propagation in fluids and/or elastic domains is performed using several available test cases. The following functionalities are available [Ref-1]:

- 3D or stick model of foundation rafts
- Shallow or embedded foundation (various shapes, rigid or flexible raft...),
- Inclined waves,
- Control points,
- Uniform and stratified soil above bedrock,
- Impedance calculations,
- SSSI,

- Option to prevent fictitious resonance
- Base mat uplift calculations

Many test-cases are available (about 30), for MISS3D and ProMISS3D, including external validation, non-regression tests, benchmarks and internal verification tests.

On an international level, the code was used in the international benchmark on the Hualien mock-up in order to model the mock-up subjected to an earthquake and so provide the transfer functions and floor response spectra. On a national level, a benchmark between SASSI 2000 and ProMISS3D was conducted between 2005 and 2007 using the test case configurations [Ref-2]. It concluded that the differences between the two codes were acceptable. Further calculations were conducted in 2010 in order to improve the level of validation in the comparison of these two codes.

In the context of the UK EPR project, ProMISS3D is used for determining the impedance functions of a shallow rigid raft.

7.4. BENEFITS AND DRAWBACKS

The linearity assumption used in the ProMISS3D approach can be considered as a simplification of the problem. However, the main benefits arise from the fact that it easily models basic physical phenomena (resonant behaviour, propagation in a homogeneous medium, far field interactions) and it can also take into account phenomena such as the diffusion of waves in random media that are currently ignored in the usual approach.

8. ROCOCO

8.1. BRIEF DESCRIPTION OF THE CODE

The ROCOCO code [Ref-1] was written so that allowable stresses for class 1 pipes could be verified.

The code comprises 3 parts, which may be used separately or in combination of two or three components, as follows:

- “THERMIQUE” option for calculation of conduction on one dimensional models
- “TITUS” option for interpretation of thermoelastic calculation results and/or pressure stresses obtained from two or three dimensional finite element analysis using the SYSTUS program (see section 12 below)
- “RCC-M” option for mechanical analysis only, and used to verify RCC-M level 0 and A criteria

The “THERMIQUE” option may be used alone or with the other two options. However, the “SYSTUS” option may only be used in conjunction with the “RCC-M” option.

8.2. CALCULATION OPTION

“THERMIQUE" option

This option performs radial conduction calculations on an axi-symmetric model, resolving the Fourier equation using a finite difference algorithm.

Boundary conditions are as follows:

- the external wall is assumed to be heat insulated (preventing any heat transfer to the outside)
- the temperature of the fluid conveyed inside the pipe and the heat transfer coefficient between the fluid and the inside wall are both specified

“TITUS” option

This option is used to re-run a file of results obtained from the finite element thermo-elastic analysis carried out using the SYSTUS code for one or more selected regions. The ROCOCO program is then used to extract the stress tensors required for the “RCC-M” option.

“RCC-M” option

This option uses the files created by the “THERMIQUE” and/or “TITUS” options. For one-dimensional analysis, the model is a straight pipe with or without geometric and/or physical discontinuities. For two-dimensional analysis, the model is almost a true representation of the structure.

This option takes account of the stress indices to be applied to the results of the “THERMIQUE” option, together with the moments due to earthquake, clearances and thermal expansion required to verify RCC-M acceptance criteria.

N.B. Stress tensors obtained from two-dimensional finite element analysis at welds are weighted by means of stress indices to allow for the effect of welding on metal properties.

N.B. For the basic design analyses, the available version of the ROCOCO code, based on RCC-M requirements is used. Nevertheless, this program will be modified to follow ETC-M requirements which are very close to RCC-M ones.

9. ROLAST-E

9.1. BRIEF DESCRIPTION OF THE CODE

The ROLAST-E code [Ref-1], which has been developed at AREVA NP GmbH, describes non-steady-state flow processes and, in particular, pressure waves in fluid-filled piping systems with an arbitrary degree of bifurcation.

The associated fluid can be any liquid, two-phase mixture, vapour or gas, with isentropic flow assumed.

9.2. MAIN FEATURES

Allowance is made for connections or bifurcation points in these sections, boundary functions, rupture discs, pumps, valves, swing check valves etc. localised at the so-called nodes of the system as well as the above shifting fluid/gas phase boundaries with the aid of special boundary conditions.

Special options are:

- pre-programmed peripheral functions for pressure or mass flow
- pumps (including pump dynamics and four-quadrant models)
- control valves
- critical flow
- non-return fittings (with/without damping)
- rupture discs
- gas bubbles with movable fluid/gas interfaces
- pressure build-up due to chemical reactions

9.3. MATHEMATICAL METHOD

The one-dimensional equations for flow of compressible media through lengths of piping of constant cross-section and wall thickness, so-called pipe sections are solved in Eulerian geometry using the method of finite differences, with computation time minimised by use of the characteristic or Lax-Wendroff method as appropriate.

9.4. VALIDATION, APPLICATION AND EXPERIENCE

ROLAST-E has been mainly validated on analytical solutions for pressure wave problems in piping systems. A description of the physical models and an input manual assist the user in the application of the code. The code has been used for analyses of the loads on steam generator divider plates and tube bundles for different pressurised water reactors.

10. S-TRAC

10.1. BRIEF DESCRIPTION OF THE CODE

S-TRAC [Ref-1] is based on the computer code TRAC-PF1, which is an advanced best-estimate computer program to calculate the transient behaviour of a pressurised water reactor. Modules from the computer code HAUPT are implemented to simulate the 2D structural behaviour under transient hydraulic forces.

S-TRAC describes the steam water flow of a hydraulic system by a full two-fluid (six-equation) hydrodynamics model, while 1D modules (e.g. pipe, tee, pump, valve, steam generator, pressuriser) as well as a 3D module (vessel) can be used in an arbitrary combination for the simulation of complex hydraulic systems.

The behaviour of the fluid and the structures are described simultaneously, i.e. there is a feedback of the induced displacement of the structures on the pressure in the surrounding fluid cells.

10.2. MAIN FEATURES

The main features of S-TRAC used for the simulation of hydraulic systems during transients are:

- Simulation of a four-component (liquid, liquid solute, vapour, non-condensable gas) two-phase (liquid, vapour) flow by eight equations
- One-dimensional flow calculation for piping systems
- Three-dimensional (r, θ, z) flow calculation for vessel type components (e.g. to allow an accurate calculation of the complex multidimensional flow patterns inside a reactor pressure vessel)
- Addition of a stratified flow regime in 1D hydrodynamics
- Flow regime dependent constitutive equations to describe the transfer of mass, energy and momentum between the steam-water phases and the thermal interaction of these phases with the system structures
- Detailed heat transfer analyses of the vessel and the loop components
- 2D (r, z) treatment of fuel-rod heat conduction with dynamic fine-mesh rezoning to simulate quench fronts movements

Consistent analysis of the entire accident sequence, i.e. the steady state calculation as well as the analyses of different phases of a transient can be performed by a single calculation.

Due to the modular structure, improvements to component modules can easily be performed without disturbing the remainder of the code.

S-TRAC currently includes the following modules: accumulator, break and fill, core, pipe, pressuriser, pump, steam generator, tee, turbine, valve, and vessel with associated internal downcomer, lower plenum, core, upper plenum.

Major aspects of the calculations are performed in separate modules (functional modularity)

10.3. MATHEMATICAL METHOD

The partial differential equations that describe the two-phase flow and the heat transfer are solved using finite differences. The heat transfer equations are treated using the semi-implicit differencing technique.

The fluid-dynamics equations in the one-dimensional components use a multi step procedure that allows the material Courant condition to be violated.

The three-dimensional vessel option uses semi-implicit differencing. The finite-difference equations for hydrodynamic phenomena form a system of coupled, non-linear equations that are solved using the Newton-Raphson iteration procedure.

10.4. DOCUMENTATION AND VALIDATION

S-TRAC is documented in code manuals. The validation of the code, which was carried out on many small and large scale experiments related to Loss of Coolant Accidents (LOCAs), is documented in many papers. AREVA NP GmbH has validated the code in particular for application to rapid transient sequences, e.g. typical blowdown scenarios.

10.5. APPLICATION AND EXPERIENCE

S-TRAC and previous versions as e.g.TRAC-PF1 have been successfully used at AREVA NP GmbH for about 25 years to analyse the hydraulic loads after LOCAs or valve opening/closing. Typical applications for pressurised water reactors were the analyses of the hydraulic loads in the RPV (especially on the control rod guide tubes in the upper plenum) and the loads on the pressuriser discharge system for different pressurised water reactors.

11. STRUDYN

11.1. BRIEF DESCRIPTION OF THE CODE

STRUDYN [Ref-1] to [Ref-3] is a general code for linear-elastic static and dynamic FE analyses of large 3D structures developed by FRAMATOME-ANP (former Siemens/KWU) to meet the actual requirements of the reactor industry. The special features of this code are its capacity for calculating large structural systems and its specialised and high-performance eigenvalue solvers and solution procedures. STRUDYN is the most frequently used tool for the seismic design of NPPs in Germany as well as of VVER-type NPPs in eastern European countries and the CIS. It has been thoroughly verified and validated through many benchmark studies by authorities around the world. The fully documented industrial software package, available in German, English and Russian runs on VAX/VMS, SGI/IRIX, HP/UNIX and DEC Alpha (open VMS) workstations and WINDOWS 2000 or XP.

11.2. MAIN FEATURES

STRUDYN is a general purpose program system for linear-elastic static and dynamic analysis of 1D, 2D and 3D structures.

The finite element library of the program contains elements suitable for the simulation of real structures.

Restraints and boundary conditions can be represented either by suppressing the pertinent nodal degree of freedom or by adding equivalent springs.

Static analysis can be performed for the following types of loads:

- Dead weight
- Concentrated loads (nodal loads)
- Distributed loads
- Pressure
- Temperature
- Imposed displacements
- Hydrostatic loads

The eigenvalue analysis is performed by a specialised Lanczos eigensolver (Lanczos-QR-method [Ref-1]). A restart capability is included. This is helpful for the analysis of large models.

Dynamic analyses are performed by means of modal analysis, harmonic analysis or the direct integration method. Modal analysis may be carried out by means of the following:

- Time-history analysis for base excitation

- Time-history analysis for transient force excitation
- Response-spectrum analysis

Response-spectrum analyses are conducted to calculate the maximum displacements and accelerations of the structure due to a support excitation by displacement or acceleration spectra for a selected number of natural frequencies and eigenvectors. Superposition of the modal responses is carried out in STRUDYN by means of 4 different algorithms, namely the absolute sum of the amplitudes (ABS) method, the absolute sum of the larger amplitude plus square root of the sums of the squares (MAXS) method, the square root of the sums of the squares (SRSS) method as well as the double sum method described in the USNRC Regulatory Guide 1.92 [Ref-2].

Data input into STRUDYN is performed via the pre-processor program STRUDOP [Ref-3] which can also read and translate existing geometry input data.

The present STRUDYN version includes the following processors and postprocessor programs:

- STRUMAT for defining system matrices
- STRUSTA for static analyses
- STRUEIG for eigenvalue analyses
- STRUMOD for determining modal values
- STRUTHM for time-history analyses
- STRURES for response-spectrum modal analyses
- DYNRES for derivation of response spectra (postprocessing of the acceleration, velocity and displacement time histories)
- STRUCOM superposes results for different load cases, e.g. static and dynamic loads
- STRUBILD plots non-deformed as well as the deformed structure

11.3. DOCUMENTATION AND VALIDATION

The basic documentation comprises the description of the mathematical principles and the software of STRUDYN and consists of the following documents:

- Theoretical manual [Ref-1]
- User information manual [Ref-2]
- Verification/validation manual [Ref-3]

The principles of the programs as well as the correctness of the whole software package have been thoroughly checked by means of a number of selected characteristic test examples.

Furthermore, the quality of the results obtained with STRUDYN has been validated by more than 1000 benchmark analyses and comparisons of results carried out by different independent experts and authorities in Germany and abroad [Ref-4] [Ref-5].

11.4. APPLICATION AND EXPERIENCE

STRUDYN has been used many times as the basic program for the analysis of the dynamic behaviour, structural response analysis and stress analysis of structures of all Siemens-type NPPs and of VVER-1000 NPPs in the CIS and eastern European countries (which were performed due to emergency and seismic loading conditions) designed by Atomenergo-project (AEP).

Investigations using STRUDYN have also been performed for a number of VVER-440 type plants in Bulgaria, Hungary and the Slovak Republic as well as for the CANDU 600-type NPP at Cernavoda in Romania.

12. SYSTAR

The SYSTAR [Ref-1] code is used to perform fatigue analyses for the verification of RCC-M Level A criteria. Notably, SYSTAR is able to account for load combinations at any instant for each situation considered, and is used to perform analyses according to RCC-M B3200.

13. SYSTUS

13.1. BRIEF DESCRIPTION OF THE CODE

SYSTUS is a software package developed by ESI Group [Ref-1] [Ref-2] for solving various mechanical, thermal and thermo-mechanical problems. SYSTUS is applied in diverse fields, including civil, mechanical, nuclear and offshore engineering, when advanced numerical techniques are required; it is a powerful research and development tool.

The modular form of SYSTUS allows great flexibility of use, and a dynamic loader tool permits the incorporation of the user's own modules such as automatic mesh generation routines or elements with special constitutive relationships.

Pre-processing:

It is necessary to distinguish two capabilities:

- a geometry and mesh pre-processor is integrated into the software environment which enables construction of different geometries (or their importation from a CAD file) and mesh generation.

- a mesh and data pre-processor: A wide range of routines allows the automatic generation of two and three dimensional meshes. The geometry, mesh, and/or the boundary conditions can be defined interactively in the pre-processing routines. Other routines can refer to, modify and add to previously defined meshes.

Post-processing:

SYSTUS contains a wide choice of routines for performing various tasks (selection, combination and other treatment of computed results). The engineer will therefore obtain results in the form best suited to his needs: e.g. plots of deformed shape, principal stresses, contour plots, and the variation of computed results over arbitrarily chosen planes or lines intersecting the structure. Fatigue and fracture mechanics problems can also be analysed. The types of problems that may be analysed using SYSTUS include non-linear and dynamic behaviour.

Structures considered for mechanical analysis may consist of beams, plates, shells or of solid spatial elements. Types of elements used for modelling are:

- plane beams or 3D beams and cables,
- straight or curved pipes,
- axisymmetrical, 2D, or 3D thin or thick shells,
- axisymmetrical, 2D, or 3D solid elements,
- fluid elements and fluid interaction elements,
- springs,
- 2D or 3D contact elements.

It is possible to perform:

- static linear or non linear analysis for various load types (forces displacements, pressure, temperature, accelerations, etc...), with use of elastic, elastoplastic, hyperelastic, viscoplastic, viscoelastic materials, and with representation of geometrical non linearities (contacts, large displacements, large rotations, large strains).
- a dynamic analysis (calculation of eigenmodes, transient analysis, harmonic analysis, seismic analysis, stochastic analysis),
- a stability analysis (linear buckling, non linear buckling and post buckling).

13.2. APPLICATION

SYSTUS is used in the detailed design of inner containment liner and the penetration sleeves in the FA3 EPR.

13.3. VALIDATION TESTS

No specific information is available at the moment.

14. SOFISTIK

14.1. BRIEF DESCRIPTION OF THE CODE

SOFiSTiK is a general purpose finite element code developed by SOFiSTiK AG. SOFiSTiK offers two programs for solving dynamic problems. The DYNA tool is utilised for linear dynamic and seismic analysis and design. ASE performs dynamic analysis by eigenvalue computation and a geometrical and material nonlinear time-step method.

The SOFiSTiK Finite Element Analysis suite also includes the following processing modules:

- AQUA - materials and cross sections
- SOFiLOAD - load generator
- CSM/CSG - construction stages in bridge and geotechnical engineering
- ASE - 3D FE Solver (linear, nonlinear, dynamic)
- DYNA - dynamic and seismic analysis
- TALPA - FE solver for geomechanics and tunnelling
- STAR - analytical beam structures
- HASE - 3D half-space and substructure analysis
- HYDRA - 3D potential analysis, seepage, hydration and fire design
- GEOS - 3D tendon geometry, pre- and post-tensioning
- MAXIMA - automatic superpositioning
- PHYSICA – multi-physics
- COLUMN - design of R/C columns
- WIST - retaining walls

14.2. MATHEMATICAL METHOD

SOFiSTiK calculates the static and dynamic effects of general loadings on a structure by dividing it into an assembly of individual elements interconnected at nodes (Finite Element Method).

In the EPR applications, the program is used for the dynamic analyses, utilising models consisting of beam, shell/plate and volume elements (additionally spring elements are used for representation of soil-structure interaction).

The numerical methods utilised by the SOFiSTiK program include consideration of effects due to non-linear material behaviour of shell structures and second and third order effects due to geometrically non-linear behaviour. Non-linear structural behaviour of reinforced concrete members only can be modelled by SOFiSTiK using shell/plate elements.

The analysis of non-linear effects in SOFiSTiK is performed iteratively using a modified Newton method with a time-dependent stiffness matrix. Non-linear material behaviour of reinforced concrete within shell/plate elements is modelled by using a layer model involving an arrangement of crosswise layers of bending reinforcement located in their correct positions near the surfaces.

The non-linear behaviour of reinforced concrete components is represented by using:

- non-linear uni-axial stress-strain laws of concrete
- tension softening of concrete after cracking dependent on fracture energy
- tri-linear stress-strain laws of reinforcing steel

14.3. VALIDATION

SOFiSTiK is validated against experimental results (MEPPEN slab tests) and by benchmarking against other software.

15. CYBERQUAKE

15.1. BRIEF DESCRIPTION OF THE CODE

Cyberquake [Ref-1] is a computer code developed by the BRGM (Bureau de Recherches Géologiques et Minières, or French Geological Survey). It can be used to study linear and non-linear behaviour of sedimentary soil columns under dynamic loading conditions (impulses, harmonic loadings, seismic loadings).

The Cyberquake computer code uses the three main types of modelling techniques used in soil dynamics:

1. Linear laws
2. Equivalent linear visco-elastic laws:

   These widely used models were presented in the early 1970s by Seed & Idriss. The models are recognised for the applicability of their results in the moderate to low seismicity context.

3. Elastoplastic laws:

   This type of computer code has been developed in recent years and offers satisfactory simulation of most soil dynamics phenomena. However, the complexity of the models used and the number of parameters needed limit their application to cases where the seismic level is high, and where detailed knowledge of the soil behaviour is available.

The Cyberquake computer code uses three modules:

1. Control Point module: This module is required when the seismic loading is defined at the free field level and needed at a lower level. It assesses the deconvolution of the time history. This module can be used for an embedded building.
2. Linear equivalent module: This module assesses the convolution of the time history using equivalent linear viscoelastic laws.
3. Elastoplastic module: This module assesses the convolution of the time history using elastoplastic laws.

15.2. METHOD

In applying the Cyberquake computer code, several modelling hypotheses are adopted:

- The soil is assumed to be laterally infinite (absence of lateral heterogeneity); definition of a one-dimensional geometrical model is thus sufficient.

- The rocky substratum is considered rigid and non-deformable. If there is a significant difference between wave propagation speeds, this could be modelled as a rigid layer. It then represents a perfect medium (infinite propagation speed). The deformable substratum is considered to be an isotropic, homogeneous, elastic material,
- The three components of the seismic signal are considered.
- The seismic motion is imposed as a time history.

15.3. QUALIFICATION

For qualification of the Cyberquake computer code, six test cases and one benchmark study have been performed as follows [Ref-1]:

Test cases:

1. Use of the Cyberquake tools

For elastic calculations:

2. Dry ground linear transient calculations
3. Linear transient calculations with groundwater

For non-linear or equivalent linear calculations:

4. Non-linear behaviour model
5. Equivalent linear behaviour model
6. Deconvolution

Benchmark study:

A case study presenting deconvolution calculations, compared EQE and EDF-SEPTEN results involving a real soil profile. Results were compared for shear module degradation with depth and distortions in the soil for two earthquakes studied in a Shake calculation by EQE and a Cyberquake calculation by EDF.

16. ANSYS

16.1. BRIEF DESCRIPTION OF THE CODE

ANSYS [Ref-1] is a software package based on the finite element method developed by ANSYS Inc. It is a general-purpose finite element software package which incorporates pre-processing (geometry creation, meshing), solver and post-processing modules using a graphical user interface.

High Fidelity Simulations

ANSYS is applicable to a broad range of structural mechanics analyses:

- Static analysis with linear or non linear stress and deformation evaluation,
- Dynamic analysis ranging from simple determination of vibration characteristics (eigenmode, harmonic or spectrum analysis in the frequency domain) to complex time-dependent transient phenomena, including dynamic effects and treatment of time-dependent material properties,
- Steady-state or transient thermal analysis that can be subsequently coupled to stress analyses.

Modelling of Complex Material Behaviour

ANSYS contains an extensive library of material models: linear and nonlinear material models for handling composites, plasticity in metals, and hyperelasticity of rubber components as well as specialised materials including cast iron, shape memory alloys, porous elasticity and cohesive zone models used to simulate debonding between parts.

Modelling of Complex Topologies

To represent complex real-world geometries, ANSYS structural mechanics solutions support a range of elements including beam, shell, solid and solid-shell elements. Also available are pretension, joints, gaskets and other special elements including nonlinear springs or smeared and discrete reinforcements.

Modelling of Interactions between Parts

Since simulated or real products usually involve multiple parts that interact, a robust and complete set of contact capabilities is available: surface–surface, line–surface and line–line contact for flexible and rigid bodies; contact behaviour for constant or orthotropic friction; and sliding behaviour for structural, thermal and multi-physical contact applications. Fast, automatic contact detection backed by powerful algorithms, allows for fast and accurate solution of models involving contacts.

16.2. APPLICATION

ANSYS V10.0 is used in the civil work detailed design of the structures below:

- internal structures of the reactor building,

- external containment,
- internal containment.

16.3. VALIDATION TESTS

No specific information is available at the moment.

17. COBEF

17.1. BRIEF DESCRIPTION OF THE CODE

COBEF:

The COBEF program [Ref-1] is a finite element code package developed by Coyne et Bellier for solving static two dimensional (plane deformation, stresses, symmetrical axis) and three dimensional problems assuming linear elasticity:

- The resolution of static problems when evaluating the response of the structure to displacements and stresses due to mechanical or thermal loading: includes calculation of reactions to support and inter-zone forces,
- Non linear components of volumes: materials under zero strain/stress or orthotropic materials or those containing micro ducts. Calculates forces in disequilibrium, displacements and stresses.

Applications used for pre- and post-processing are:

- MAILEF: Generation of meshes for structures,
- TRACEF3: Graphic tracing of meshes and deformations,
- TRASIG3D: Graphic tracing of meshes and stresses in three dimensions.

MAILEF (pre-processor)**:**

MAILEF is a standard pre-processor within the COBEF finite element program suite. Its purpose is to generate files containing data required by the program (mesh, boundary conditions, material properties ...).

TRACEF3 (Three-dimensional Mesh Generation) (pre- and post-processor)**:**

TRACEF3 is a program which enables the user to generate meshes and deformed meshes for structures processed by COBEF.

TRASIG3D (post-processor)**:**

The TRASIG3D program is a post-processor used with COBEF which is used to display stress states and isovalues on user defined surfaces comprising the faces of the elements. The shape module enables the user to display, draw, and check three dimensional meshes (surface contours, element contours). The program is interactive (but has an option for handing control to a command file). **TRASIG3D** was developed on the basis of the interactivity concepts also applied in **TRACEF3**, many of which have been enhanced and extended.

FERRAIL:

FERRAIL is designed to evaluate steel reinforcement in a plate or shell loaded by external forces. Steel reinforcement layers are represented as lying orthogonally in the upper and lower sides of the plate. Forces are input from a file generated by COBEF. The steel reinforcement calculation may be performed according to rules from different design codes (EC2, ETC-C, ACI) depending on the sub-version used.

17.2. APPLICATION OF COBEF

COBEF (version 4.1) is used for the civil works detailed design of the following FA3 EPR structure:

- the Nuclear Auxiliary Building,

COBEF facilities which have been used in the FA3 EPR design have been limited to linear static structural analysis to facilitate structural design calculations.

These facilities are linked to the standard pre- and post-processors of COBEF such as:

- those which are part of the **PRECONT** or **FERRAIL** programs for steel reinforcement design,
- other processors such as the **MAILEF** mesh generator and graphic output post-processors such as **TRACEF3** and **TRASIG3D**.

17.3. VALIDATION TESTS

For each element (above), qualification tests have been carried out, and results compared with information published in the technical literature.

A selection of validation studies performed are listed below:

- **CESA Project** : EDF Maeva mock up – Comparison between predictive calculations and experimental results –COBEF calculations were compared with results obtained by others organisation (CEA, IPSN, ISMES…),
- **TACIS Project** : evaluation of the prestress levels in VVER 1000 Unified and Small series containments (Russian Federation),
- **Stress Assessment of P4n P’4 and N4 containments**: EDF project conducted between 1998 and 2005. Results were submitted to the French authorities (now ASN) and are summarised in Report DES458 reviewed by the French GPR.
- **Tests on “parallelogram” shaped plate model using shell elements – Comparison of COBEF and ANSYS :** The AFNOR validation guide for structural analysis codes (Guide de Validation des Progiciels de Calcul de Structures) proposes a test involving a “parallelogram” shaped plate model.

These validation tests have confirmed that COBEF version 4.1 is an acceptable tool for structural calculations in nuclear projects.

18. HERCULE

18.1. BRIEF DESCRIPTION OF THE CODE

The computer code HERCULE [Ref-1] is a structural analysis code package applicable to civil engineering problems. The code package consists of a set of program modules capable of solving a range of civil engineering complex tasks.

HERCULE is a FORTRAN IV program that was designed and developed by SOCOTEC.

Features of HERCULE are as follows:

- Contains a complete element library which is regularly updated (truss members, beams with end releases and eccentricities, triangular and quadrilateral membranes, plate and shell elements with different shape functions, linear and iso-parametric volume elements, cable elements),
- Full element compatibility with automatic elimination of unconnected degrees of freedom,
- Complete modularity using free format language for automatic mesh generation,
- Ability to check data before processing with user defined graphic outputs,
- Complete dynamic and stability analysis capabilities using static modelling of the structure,
- Optimised algorithm which minimises the size of the central memory allocated and the execution speed,
- Dynamic memory allocation allowing optimised use of the computer memory,
- Printing independent from the calculation,
- Various graphical displays of the results available, including projections, deflected curves, isovalues, etc,
- Ability to take into account any connection which can be defined by a set of linear equations that link its degrees of freedom.

18.2. APPLICATION

HERCULE is used in the civil works detailed design of the following structures:

- Nuclear island raft,
- Airplane crash shield,
- Fuel building,
- Electrical building,

- Diesel building,
- Access building.

18.3. VALIDATION TESTS [REF-1]

HERCULE has been validated against twenty-four validation tests provided by the code developer SOCOTEC. Results confirm that HERCULE version 34.01 is an appropriate tool for structural calculations in nuclear projects.

19. NASTRAN

19.1. BRIEF DESCRIPTION OF THE CODE

NASTRAN (NASA Structural Analysis Program) [Ref-1] [Ref-2] is a finite element analysis code for simulating stress, dynamics and vibration in real-world, complex systems. NASTRAN represents a package of software products which originated in the aerospace industry but are now extensively used in mechanical, consumer, automotive, and civil engineering. Versions of NASTRAN are marketed by different software vendors containing different proprietary features. Different versions of the NASTRAN available include MSC. NASTRAN, NE NASTRAN and xMG NASTRAN.

Structural and modelling elements are provided for the representation of the more common types of structural building blocks, including rods, beams, shear panels, plates, and surfaces of revolution. More general types of building blocks can be created by combining these simple elements or by using a "general" element capability. The sub-structuring capability allows different sections of a structure to be modelled jointly after having already been modelled individually.

Capabilities include:

- Strength, durability and vibration assessment of structures,
- Structural dynamic response simulation of loads that vary with time or frequency,
- Static and transient analysis of structures involving nonlinearities,
- Heat transfer analysis with contact including conduction, convection and radiation,
- Failure models based on virtual crack closure techniques and cohesive elements,
- Effects of aero elasticity on structures,
- Combined topology, sizing and shape optimisation,
- Optimise large model designed sections through automatic external super-elements,
- Enhanced iterative and in-core sparse solvers.

NASPOST (Version 7.02):

NASPOST is a code developed by Sir Robert McAlpine Ltd (SRM) for pre- and post-processing. The program provides a graphical user interface and menu driven pre- and post-processor for use with the proprietary NASTRAN software. IN the FA3 design, NASPOST is used as pre processor for modifying FE meshes used by COBEF for the Nuclear Auxiliary Buildings and for visualisation of results from local NASTRAN FE analyses. The program is capable of interfacing with both the MSC and UAI versions of NASTRAN.

PATRAN:

PATRAN is a widely used pre- and post-processing program for Finite Element Analysis (FEA), providing solids modelling, meshing, and analysis setup for MSC NASTRAN, ANSYS and other software.

19.2. APPLICATION

The code NASTRAN is used in the civil work detailed design for the FA3 Nuclear Auxiliary Building.

19.3. VALIDATION

No specific information is available at this time.

APPENDIX 3 – REFERENCES

External references are identified within this sub-chapter by the text [Ref-1], [Ref-2], etc at the appropriate point within the sub-chapter. These references are listed here under the heading of the section or sub-section in which they are quoted.

1. ASTER

1.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] Description of the code (http://www.code-aster.org). (E)

[Ref-2] Qualification:ENGSDS080130 A and A0.02.047 A
(available at http://www.code-aster.org) (E)

2. EUROPLEXUS

2.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] Description of the code (http://europlexus.jrc.ec.europa.eu/). (E)

3. JUPITER

[Ref-1] I Duvernoy. JUPITER qualification summary report. NFPMR DC 159 Revision A. AREVA. July 2005. (E)

[Ref-2] O Ibidapo-Obe, A Sofoluwe, O Abass. Householder/QL algorithm for the functional eigenproblem. Advances in Engineering software, volume 4, issue 4. October 1982. (E)

[Ref-3] R J Guyan. Reduction of stiffness and mass matrices AIAA Journal, vol. 3(2). 1965. (E)

4. MAGIC

4.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] MAGIC software version 4.1.1: Mathematical model HI-82/04/024 Revision B. EDF-SA. 2005. (E)

4.3. VALIDATION

[Ref-1] Qualification file of fire code MAGIC version 4.1.1. HI-82/04/022 Revision B. EDF. December 2005. (E)

6. PRECONT – COMPUTER ASSISTED ANALYSIS AND DESIGN OF PRE-STRESSED CONCRETE CONTAINMENTS.

6.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] PRECONT qualification documents and users guide are available via EDF at COB office on demand.

APPENDIX 3A.6 - TABLE 1

[Ref-1] Allong Module: Calculation of cable elongation during tensioning. 60590011-UM-008 Revision A. October 1997. (E)

[Ref-2] Calpdom Module: Decomposition of dome cables into straight segments and circular arcs. 09456010-RP-004 Revision A. October 2008. (E)

[Ref-3] Coth Module: Calculation of intersection point elevations of horizontal cables with regularly spaced radial planes. 09456010-RP-001 Revision A. October 2008. (E)

[Ref-4] Cotv Module: Calculation of vertical cable spacing at a given elevation. 09456010-RP-002 Revision A. October 2008. (E)

[Ref-5] Cotdom Module: Calculation of elevation of a dome cable at the intersection with a cylinder with a given radius. 09456010-RP-003 Revision A. October 2010. (E)

[Ref-6] Defcab Module: Encoding of horizontal and vertical cables (original version replaced by EDICAB). 09456010-RP-007 Revision A. October 2010. (E)

[Ref-7] Defcom Module: Encoding and discretisation of dome cables (original version replaced by EDICAB). 09456010-RP-005 Revision A. October 2010. (E)

[Ref-8] Destens Module: Calculation and design of cable tension as a function of curvilinear abscissa. 60590011-UM-007 Revision A. October 1997. (E)

[Ref-9] Discr Module: Discretisation of an encoded horizontal or vertical cable (see EDICAB). 60590011-UM-007 Revision A. October 1997. (E)

[Ref-10] Discrdom Module: Discretisation of encoded dome (using DEFDOM). 09456010-RP-006 Revision A. October 2008. (E)

[Ref-11] Ecartdom/distdom Module: Calculation of spacing (on a plane) between points of intersection of a dome cable layer with cylinder of a given radius. 09456010-RP-008 Revision A. October 2008. (E)

[Ref-12] Edicab Module: Encoding and discretisation of all types of cables with graphic aid (replaces DEFCAB, DEFDOM, DISCR and DISCRDOM).
60590011-UM-002 Revision B. February 1998. (E)

[Ref-13] Fnodal Module: Calculation of vector loading due to pre-stressing applied to finite element model. 60590011-UM-006 Revision A. December 1997. (E)

[Ref-14] Mistens Module: Calculation of cable tension as a function of curvilinear abscissa. 09456010-RP-009 Revision A. October 2008. (E)

[Ref-15] Recol Module: Re-fusing of vertical and gamma cables. 09456010-RP-010 Revision A. October 2008. (E)

7. PROMISS3D

7.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] Features ProMISS3D (http://www.mssmat.ecp.fr/structures/perso/miss) (E)

[Ref-2] Quality sheet and qualification file ProMISS 3D V1.3 and chaining with Code_Aster version 7.
H-T62-05-009 Revision A1. EDF-SA. July 2009. (E)

H-T62-05-009 Revision A1 is the English translation of H-T62-05-009 Revision A.

7.3. VALIDATION

[Ref-1] Quality sheet and qualification file for ProMISS 3D version 1.4 and its chaining with Code_Aster version 9.
H-T62-2009-03724-EN Revision 1.0. EDF-SA. April 2012. (E)

H-T62-2009-03724-EN Revision 1.0 is the English translation of H-T62-2009-03724-FR Revision 1.0.

[Ref-2] Validation of calculation tools used in soil-structure interaction – Summary Report.
30-05/11 Revision A1. August 2010. (E)

8. ROCOCO

8.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] S Lame. ROCOCO Programme Qualification Summary Report.
PEER-F DC 89 Revision A FIN. AREVA. October 2012. (E)

9. ROLAST-E

9.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] G Hilbert. Validation of ROLAST. NEPR-G/2007/en/0122 Revision A. AREVA. 2007. (E)

10. S-TRAC

10.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] Validation of S-TRAC. NGPS1/2005/en/0076. AREVA. 2005. (E)

11. STRUDYN

11.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] STRUDYN. Theoretical Manual. Version 1/1996. (E)

[Ref-2] STRUDYN. User Information Manual. Version 1/2000. (E)

[Ref-3] STRUDYN. Verification Manual. Part 1 – 5, 1/1996. (E)

11.2. MAIN FEATURES

[Ref-1] STRUDYN. Theoretical Manual. Version 1/1996. (E)

[Ref-2] Regulatory Guide 1.92 Revision 2. USNRC. July 2006. (E)

[Ref-3] Benchmark for the computer code STRUDYN (Quality Assurance) Part 1. KWU-Arbeitsbericht R621/202/82. (E)

11.3. DOCUMENTATION AND VALIDATION

[Ref-1] Theoretical Manual. STRUDYN. Version 1/1996. (E)

[Ref-2] User Information Manual. STRUDYN. Version 1/2000. (E)

[Ref-3] Verification Manual. STRUDYN. Part 1 – 5, 1/1996. (E)

[Ref-4] Benchmark for the computer code STRUDYN (Quality Assurance) Part 1. KWU-Arbeitsbericht R621/202/82. (E)

[Ref-5] Benchmark for the computer code STRUDYN (Quality Assurance) Part 2. KWU-Arbeitsbericht R621/83/0166. (E)

12. SYSTAR

[Ref-1] S Lame. SYSTAR Programme Qualification Summary Report. PEER-F DC 88 Revision A FIN. AREVA. October 2012. (E)

13. SYSTUS

13.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] ESI-group web site – description of the code. (http://www.esi-group.com/products/multiphysics/systus). (E)

[Ref-2] S. Courtin. Qualification synthesis report of SYSTUS computer code, integrating SYS* tools, the F.E. CRACK BLOCK and the NUKE module. NFPMR DC 68 Revision E. AREVA. April 2008. (E)

15. CYBERQUAKE

15.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] P A Naze. Principles of the CYBERQUAKE computer code. E-N-S-GG/00-00355 Revision A1. EDF. May 2000. (E)

15.3. QUALIFICATION

[Ref-1] P A Naze. CYBERQUAKE computer code validation report. E-N-S-GC/00-00356 Revision A1. EDF. May 2000. (E)

16. ANSYS

16.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] Description of the code (http://www.ansys.com). (E)

17. COBEF

17.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] Description of the code (http://www.coyne-et-bellier.fr/dun/dsc/logiciels.html). (E)

18. HERCULE

18.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] S Cazadieu. Hercule Computer Code Description. 0500 NT 001 Revision A. IOSIS. May 2009. (E)

18.3. VALIDATION TESTS

[Ref-1] S Cazadieu. Hercule Finite Element System – Validation Tests. 0500 NT 002 Revision A. IOSIS. May 2009. (E)

19. NASTRAN

19.1. BRIEF DESCRIPTION OF THE CODE

[Ref-1] Description of the code (http://www.mscsoftware.com). (E)

[Ref-2] Complement for the description of the code (http://www.openchannelfoundation.org/projects/NASTRAN) (E)