PRONET Update - 19

CODE aster

QUARTERLY REPORT OF CODE aster PROFESSIONAL NETWORK

The new salome_MECA 2022 under LGPL license is available for download.

2022 version integrates SALOME 9.8

CODE_aster 16.2

CODE aster professional network
user community of
CODE aster and salome_meca

Information content:

- Open source and ProNet
- CODE aster as a research platform
- CODE aster as an industrial platform
- CODE aster as an educational platform
- CODE aster for service providers

YouTube   LinkedIn

Jean-Raymond Lévesque
Sylvie Courtier-Arnoux
Representatives of CODE aster ProNet
contact@code-aster-pronet.org

New members 2022

UKRAINE

E.O. Paton Welding Inst

New members 2023

FRANCE

FRAMATOME

framatome

ITALY

MATHEMATICAL METHODS and MODELS for ENGINEERING

Active members in 23 countries in the world

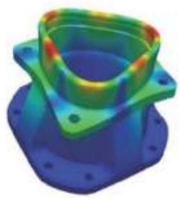
TRAINING

***Normally several training sessions for* CODE aster *and* salome_meca *are proposed each year,* but in the context of the COVID 19 pandemic, some adaptations were necessary: please contact the different teams directly.**

PHIMECA	Alter Ego	Deltacad
www.phimeca.com/Formations	www.aego.ai/training	www.code-aster-services.org

SimulEase	ing. Fondation dell'ordine degli ingegneri della provincia di Milano	aeroengineering.co.id aeroengineering services in Indonesia
contact@simulease.com	info@foim.org	www.services.aeroengineering.co.id [2]

	code_aster CLOUD_HPC CODE_ASTER Come utilizzare Code_Aster sul cloud per migliorarne la scalabilità (servizio offerto da CFD FEA SERVICE) Venerdì 16 Dicembre ore 16:00 In diretta su twitch canale FGCAEANALYST	FG CAE ANALYST
scopeingenieria@gmail.com	fgcaeanalyst@gmail.com	

Technical Courses Cursos Técnicos **Modelling, Meshing and Postprocessing with Salome-Meca Course** www.technicalcourses.net	 + Engineering simulations with open source codes – Dr. Franco Concli Free University of Bozen-Bolzano

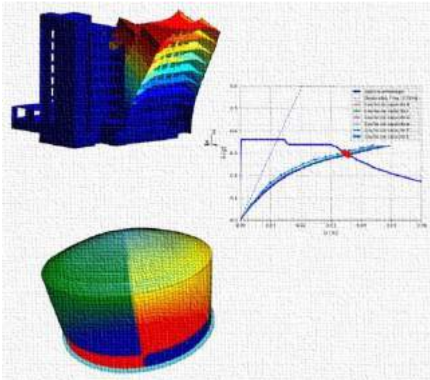
TRAINING IN SEISMIC CALCULATIONS OF STRUCTURES
STANDARD AND ADVANCED METHODS

April 5, 2023

The EUROCODES provide engineers with a general framework for justifying structures under earthquakes and envisage the use of advanced methods (push-over, non-linear transient calculations, etc.) without however giving the practical elements for their implementation.

Based on a case study (building), the training will review the different calculation approaches, from the simplest to the most complex, emphasizing the practical aspects of their implementation and the expected structural optimizations.

sixense necs



Patronas code_aster

Themes

- Theoretical and regulatory notions on seismic calculation.
- Optimization of structures subjected to the earthquake.
- Standard and advanced numerical calculations.
- Case studies and functionalities of finite element analyzes (PATRONAS and Code_Aster).

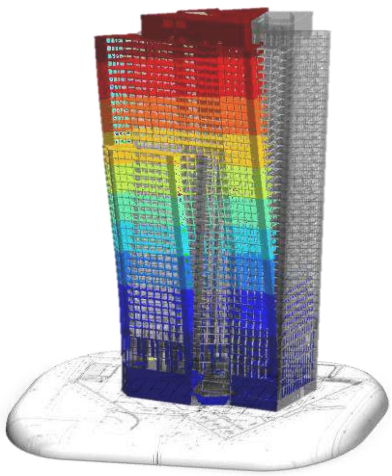
*

Educational goals

- In-depth knowledge of the principles of spectral modal analysis,
- Understanding of the basic principles of push-over analyzes (theoretical and practical notions),
- Understanding of the basic principles of non-linear transient analysis for seismic calculations,
- Through the case study and feedback:
  - analysis of the possibilities and performance of the various methods,
  - presentation of the rules of the art that allow these calculations to be carried out (checks, orders of sizes, pitfalls to avoid, etc.)

Prerequisites

- Concept of the fundamentals of dynamics and in particular spectral modal analysis,
- Experience in the practice of seismic calculations using finite element software,



Targeted audience

- Structural engineers from design offices wishing to deepen and update their knowledge,
- Project managers participating in the construction of structures subject to severe earthquakes (in particular in nuclear field),
- Operators of infrastructures subjected to severe earthquakes and prescribers of this type of advanced methods (nuclear, industry, etc.).

Comparison of Code_Aster to Abaqus usage and accuracy of the FEA analysis

Nick ANANCHENKO, Oleh MAKHNENKO

E. O. Paton Electric Welding Institute – Mathematical Modeling Department – Ukraine

In this project, bridge orthotropic deck stress-strain analysis was made to predict its deflection and assess structural integrity. The problem of large deck deflection leads up to a cracking of the bridge's asphalt surface layer which results in costly annually repairs. In order to get rid of the large deflection, strengthening repair of the bridge deck was designed, by welding the reinforcing plate to the orthotropic deck plate. Orthotropic deck FEA analysis was made in both **Code_Aster** and Abaqus Learning Edition, therefore difference between two obtained solutions was analyzed along with its accuracy.

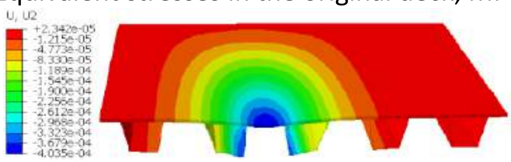
Section between two longitudinal main beams and two transverse beams was analyzed, load from a car wheel (50 kN) was applied between two rib stiffeners to obtain a maximum value of out-of-plane deck deflection. For saving the computational recourses and in order to build a fine mesh, a quarter of section was considered at symmetrical conditions.

Results obtained for original (not reinforced) orthotropic deck with **Code_Aster** show that maximum out-of-plane deflections (1.2 mm) and equivalent von Mises stresses (110 MPa) occur in the local area of applied load, as well as transverse (+119/-119 MPa) and longitudinal (+70/-76 MPa) stress components.

At the upper surface, in the area of applied load, transverse and longitudinal stresses are compressive, at the lower surface stresses are tensile. Unlikely, maximum compressive through-thickness stresses occur in the walls of rib stiffeners. FEA analysis has shown that for orthotropic deck, reinforced with an additional plate, out-of-plane deflection decreases to 0.4 mm, and equivalent stresses decrease to 23 MPa.

Deflection in the original deck, mm

Equivalent stresses in the original deck, MPa



Deflection in the reinforced deck, mm

Due to the local nature of loading, deflections and stresses occur at the local area of applied load, where finite element method cannot guarantee an exact solution, therefore several parameters of FEA analysis were varied in order to achieve accurate results. Solutions obtained in both **Code_Aster** and Abaqus were compared for one, two and three number of elements throughout the thickness and for linear vs quadratic finite element shape functions. Linear finite element usage has led to unrealistic results, as obtained this way stresses and deflections are three times lower than the real ones.

It can be explained by the local bending of the deck plate, and its thin walled structure. Also, while increasing the number of elements throughout the thickness, stresses in **Code_Aster** converged with up to 5% accuracy. In Abaqus, the solution converged (up to 1%), regardless of the number of elements throughout the thickness. Nevertheless, solution for three elements throughout the thickness and a quadratic type of finite elements did not completely converge for **Code_Aster** and Abaqus, giving 4% difference in obtained deflection, and 8% difference in obtained stresses. Also, direct method of solving the governing equations was compared to an iterative.

For both Abaqus and **Code_Aster** direct and iterative methods gave the same results, with less than 1% difference.

Mesh			**Code_Aster**						**Abaqus**				
Nb elem	Type		δ, mm	σ trans, MPa	σ thick, MPa	σ longt, MPa	σ mises, MPa		δ, mm	σ trans, MPa	σ thick, MPa	σ longt, MPa	σ mises, MPa
1	quad		1,2	97/-100	-69	55/-65	92		1,25	139/-141	-82	88/99	120
2	quad		1,2	113/-114	-68	66/-73	104		1,25	139/-141	-77	88/-99	120
3	quad		1,2	119/-119	-68	70/-76	110		1,25	140/-142	-81	90/-99	120
1	lin		0,42	21/-23	28*	16/-22	27		6,5	0	-10	25	24
2	lin		0,4	28/-16	41*	20/-20	34		1,02	36/-37	-15	22/-22	30
3	lin		0,38	31/-17	50*	23/-22	38		0,87	33/-34	-9	23/-23	30

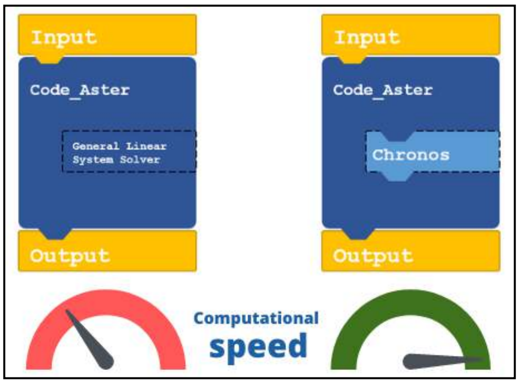
Special thanks to **Jacques Berthellemy** from **Cerema** for sharing his wide experience of bridge design with me.

Would you like to make Code_Aster faster?

D. COLOMBO, N. TARDIEU, M. FRIGO, C. JANNA

IFP Energies Nouvelles – France EDF – France - M3E – Italy

The numerical simulation of structural mechanics applications via finite elements usually requires the solution of large-size linear systems, especially when accurate results are sought for derived variables, like stress or deformation fields. Such a task represents the most time-consuming kernel in a simulation and motivates the development of robust and efficient linear solvers for these applications. On the one hand, direct solvers (such as MUMPS) are traditionally used in computational mechanics due to their robustness and ease of use, but their computational complexity and low scalability limits their applicability if the problem size increases. On the other hand, iterative solvers (such the ones available in PETSc library) offer a higher degree of parallelism and scalability especially for 3D problems but require more knowledge from the user for an efficient setup, and convergence is not always guaranteed, especially in ill-conditioned problems.



This paper presents Chronos [1], a library for High Performance Computing specifically designed to accelerate the iterative solution of linear systems within **Code_Aster**, guaranteeing an outstanding robustness and efficiency even with challenging real-world simulations and without any fine tuning of the default parameters.

Chronos provides a collection of preconditioners based on Algebraic MultiGrid (AMG) and approximate inverses along with the most popular iterative methods such as PCG, GMRES, BiCGstab. The user can easily tune its solver through a simplified interface reaching performance unachievable with a direct solver and allowing for convergence also in ill-conditioned problems where other packages based on iterative methods may fail. Thanks to its structure, Chronos can be extremely easily linked to any Code_Aster version, without changes in the workflow and in the computed results.

In the following sections we present the performance improvements achieved by Chronos embedded into Code_Aster and used as iterative linear solver. The evaluation of the computational performances has been performed on two real-case applications, selected by IFPEN and EDF, among their most interesting and challenging problems.

'Code_Aster & Chronos' for reservoir and basin modeling

At IFPEN Code_Aster is used for hydromechanical simulations in porous media at both reservoir and sedimentary basin scales. These simulations are carried out by coupling one of the fluid flow codes developed by IFPEN with a modified version of **Code_Aster**. The two coupled codes use the same mesh built from a geological model by means of classical reservoir or basin modeling tools. This mesh is well adapted to the finite volume method used in the fluid flow codes but is problematic for the finite element method used in **Code_Aster**.

Reservoir and basin models exhibit geological features like thin sedimentary layers and stratigraphic pinch-outs[i] which makes it difficult to generate a good quality mesh, even at the cost of enormously increasing the number of elements in the mesh. Consequently, bad aspect ratios and distorted elements are always present in geological meshes. These elements could have detrimental effects on the precision of the finite element solution, and, in some cases, they could make the finite element solver fail due to ill-conditioning.

To reduce these problems, the S-FEM method has been implemented in a prototype version of **Code_Aster** used by IFPEN [2]. If on the one hand S-FEM has dramatically reduced negative effects of problematic elements, on the other hand it has increased the solver calculation time: the stiffness matrix produced by S-FEM is less sparse than the one produced by FEM. In order to reduce solver time without losing robustness, Chronos has been tested on several industrial basin models and its performance has been compared with MUMPS direct solver and PETSc iterative solver available in Code_Aster (with default set-up options).

First, we present the results obtained for two elastic models with about 750.000 and 3.3 million dofs, respectively. From table 1 we see that Chronos is significantly faster than PETSc and MUMPS. PETSc is faster than MUMPS only for the 3.3 M dof model, showing a poor performance for the smaller model due to the very bad mesh quality of this case (presence of very thin and curved sedimentary layers with abrupt changes in layer depth).

	750.000 dof model 32 CPUs		3.3 M dof model 216 CPUs
	FEM	**S-FEM**	**FEM**
MUMPS	10 s	30 s	273 s
PetSC	19 s	30 s	101 s
Chronos	2.5 s	7 s	6 s

Table 1 – Time in seconds for the solution of the discretized linear system from basin modeling.

Chronos performance has been finally tested on a full hydromechanical simulation of the Neuquen sedimentary basin model [3]. The model is composed by 315.000 dof (figure 1). Results reported in table 2 clearly show the ability of Chronos to manage problematic meshes while significantly increasing performance, especially for the more computationally expensive S-FEM model.

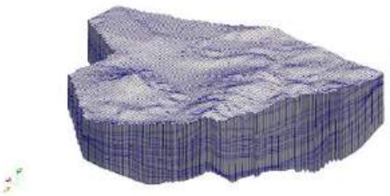


Figure 1 – Mesh of the Neuquen basin model [3] used for the hydromechanical simulations (vertical scale exaggeration: 10x). Thin layers and pinch-outs are clearly visible.

	FEM (hh:mm)	S-FEM (hh:mm)
MUMPS	6:30	30:00
Chronos	2:35 (-61%)	6:00 (-80%)

Table 2 – Total simulation time for the Neuquen basin model (315.000 dof) [3] on 72 CPUs, including time for actions not correlated to the mechanical solver (e.g. fluid flow simulation, mesh generation, data exchange between coupled codes).

'Code_Aster & Chronos' for a nuclear reactor

The second application used to evaluate the performance of Chronos has been provided by EDF R&D, where the mechanical properties of the containment building of a nuclear reactor are investigated. This building protects both the reactor from external aggressions and the environment if an internal accident occurs. Robust and accurate numerical simulations are thus required for both design and safety analysis. We consider a complex mechanical model, including several kinds of finite elements (FE): 3D continuum mechanics FE for the concrete, 1D FE for the prestressing cables, 2D FE for the internal metallic liner. Considering the prestressing cables makes the mathematical problem even more difficult, since the cables ends are attached to the concrete nodes using Lagrange multipliers to enforce the kinematic constraint. Due to the mix of different FE models and to the very high number of Lagrange multipliers, the underlying linear system is a real challenge for iterative solvers and direct solvers are often the fallback solution (see [4] for further details).

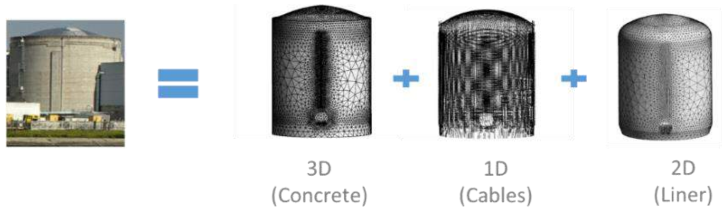


Figure 2 – Model of a containment building (courtesy of EDF R&D)

For this model, we consider three levels of refinement of the domain, X1, X4 and X8, whose number of unknowns is reported in table 3 along with the computational resources used. The number of CPUs used for each refinement level is the minimum number of CPUs allowing to MUMPS to store data in memory without swapping on disk.

	FEM X1 442,75 dof 32 CPUs	FEM X4 5.5 dof 64 CPU	FEM-X8 31.7 M dof 256 CPU
MUMPS	84 s	1079 s	> 8 h
Chronos	31 s	257 s	1372 s

Table 3 – Time in seconds for the solution of the discretized linear system of the containment building

As shown in table 3, Chronos outperforms MUMPS with the speed-up increasing with the problem size, that is 2.64 for X1, 4.19 for X4 and more than 20 for X8. In the X8 case, MUMPS is not able to complete the factorization within 8 hours, whereas Chronos requires only about 22 minutes although the problem is severely ill-conditioned.

Conclusions

'**Code_Aster** & Chronos' showed excellent results in terms of computational speed-up with the two above presented applications. We are constantly looking for large and challenging simulations, to enlarge its successful track record: get in contact with us if you want to assess the performance of Chronos in your long-lasting and tedious simulations!

References

[1] Chronos web page: https://www.m3eweb.it/chronos/

[2] Daniele Colombo, Slah Drira, Ralf Frotscher, Manfred Staat, *An element-based formulation for ES-FEM and FS-FEM models for implementation in standard solid mechanics finite element codes for 2D and 3D static analysis*, Int J Numer Methods Eng. 2022; 1-32. doi: 10.1002/nme.7126

[3] Berthelon J., Bruch A., Colombo D., Frey J., Traby R., Bouziat A., Cacas-Stentz M.C., Cornu T., *Impact of tectonic shortening on fluid overpressure in petroleum system modelling: insights from the Neuquén basin, Argentina*, Marine and Petroleum Geology, 127 (2021), 104933. doi: 10.1016/j.marpetgeo.2021.104933

[4] Carola Kruse, Vincent Darrigrand, Nicolas Tardieu, Mario Arioli, Ulrich Rüde. *Application of an iterative Golub-Kahan algorithm to structural mechanics problems with multi-point constraints*. Adv. Model. and Simul. in Eng. Sci. 7, 45 (2020).

Code_aster / Salome_meca EDF users club May 19, 2022

Guilhem FERTE and alii

EDF R&D. Department – France

CODE aster

NEW CODE_ASTER 16.2

New Materials, New Loadouts

- Plane wave loading for heterogeneous or undamped media.
- New command of homogenization CALC_MATE_HOMO for the calculation of the equivalent elastic parameters of a periodic structure with base cells, 3d or plate.
- Introduction of the nonlinear plastic / viscoelastic model KICHENIN_NL for the modeling of HDPE.
- Mohr-Coulomb law for the simulation of geotechnical structures.
- Maxwell's viscoelastic model.
- Development of a new behavioral model for soil mechanics KH_SSM based on modified CLAM-CLAY.

Parallelism, python API

- New operator massively HPC MECA_NON_LINE for the nonlinear mechanical analyzes HPC (not qualified). Current scope: large and small elastoplastic deformations.
- Improvement of the parallel performances of calculation of the potential energy (POST_ELEM).
- Binding python with pybind11 instead of boost.
- New methods of manipulation of the matrices, second members and computations of elementary terms in python.
- New methods for manipulating fields in python.

New features, new analytics

- Extension of the functionality of the CALC_FERRAILLAGE and COMBINAISON_FERRAILLAGE dimensioning operator for civil engineering, now applicable to 1D elements of the beam and column type
- Rewrote the matching algorithm for contact.
- More robust resolution for the elements DIS_CHOC and DIS_CONTACT.
- Non-symmetric stiffness matrices for the discrete elements.
- New operator POST_JMOD of the integral J modified for the analysis at rupture in the presence of initial deformations and non-proportional loadings.

Verification and validation

- Benchmark code_aster / abaqus for calculations of polycrystalline test pieces.

salome MECA

SALOME-MECA 2022 OUTING

Test release 2022.0

- Release scheduled for 09/30/2022 (publication on the website)
- Native on Scibian9
- Singularity container for other OS
- Available on cronos for use via exceed
- Need for a native version on scibian 10?
- Based on salome 9.9 a priori, salome 9.8 at least
- Contains code_aster 15.6 (stable, qualified) and 16.2 (testing)
- LGPL version container: published on the same date on the website

Final version 2022.1

- Release scheduled for 12/15/2022 (as a Debian package)
- Qualified on the Scibian9 reference platform
- Same method of distribution as before for the rest

International Workshop on Numerical Software Verification (2017)

Studying the Numerical Quality of an Industrial Computing Code:
A Case Study on Code_aster

François FEVOTTE (TRISCALE Innov)
Bruno LATHUILIERE (KMW + Nexter Defense Systems)

We present in this paper a process which is suitable for the complete analysis of the numerical quality of a large industrial scientific computing **Code_aster**.

Random rounding, using the Verrou diagnostics tool, is first used to evaluate the numerical stability, and locate the origin of errors in the source code. Once a small code part is identified as unstable, it can be isolated and studied using higher precision computations and interval arithmetic to compute guaranteed reference results.

An alternative implementation of this unstable algorithm is then proposed and experimentally evaluated. Finally, error bounds are given for the proposed algorithm, and the effectiveness of the proposed corrections is assessed in the computing **Code_aster.**

Computational Mechanics (2022)

A fully non-invasive hybrid IGA/FEM scheme for the analysis
of localized non-linear phenomena

Evgeniia LAPINA, Paul OUMAZIZ, Robin BOUCLIER & Jean-Charles PASSIEUX
(Institut Clément ADER - INSA de Toulouse)

This work undertakes to combine the interests of IsoGeometric Analysis (IGA) and standard Finite Element Methods (FEM) for the global/local simulation of structures.

The idea is to adopt a hybrid global-IGA/local-FEM modeling, thereby benefiting from:

(i) the superior geometric description and per-Degree-Of-Freedom accuracy of IGA for capturing global, regular responses,

(ii) (ii) the ability of FEM to compute local, strongly non-linear, or even singular behaviors. For the sake of minimizing the implementation effort, we develop a coupling scheme that is fully non-invasive in the sense that the initial global spline model to be enriched is never modified and the construction of the coupling operators can be performed using conventional FE packages.

The key ingredient is to express the FEM-to-IGA bridge, based on Bézier extraction, to transform the initial global spline interface into a FE one on which the local FE mesh can be constructed. This allows to resort to classic FE trace operators to implement the coupling. It results in a strategy that offers the opportunity to simply couple an isogeometric code with any robust FE code suitable for the modelling of complex local behaviors. The method also easily extends in case the users only have at their disposal FE codes.

This is the situation that is considered for the numerical illustrations. More precisely, we only make use of the FE industrial software **Code_aster** to perform efficiently and accurately the hybrid global-IGA/local-FEM simulation of structures subjected locally to cracks, contact, friction and delamination.

Documentatiion Code_aster / Salome_meca published in Italian

Francesco GRISPO and alii

Code_Aster in Italia – Italy

CODE_ASTER ITALIA

La community italiana di Code_Aster e del CAE Opensource

Documentazione

To date, the **Code_Aster** documentation is present only in French and English.

Wanting to expand the use of this software, in **Code_Aster Italia** we are trying to translate all the documents present on the official site and subjected to the GNU license from French into Italian.

The translation will be divided into 3 steps:

1) Translation of all documents automatically using google translate documents. (GT)
2) Manual translation of documents with major inconsistencies. (GTR)
3) Review of the documents that will be reported. (R.XX)

Each document will be accompanied by an indication indicating the revision status it is in.

In the studio, on the other hand, you will find simplified handouts that will guide you on setting up the analyses, explaining the various processes in a simplified way.

It will take a long time to complete each section, but it will certainly make a significant contribution to the growth of the Italian community.

By the way, if you want to contribute, write to us at fgcaeanalyst@gmail.com


Suède
Finlande
Norvège
Royaume-Uni
Irlande
Biélorussie
Pologne
Ukraine
Roumanie
Espagne
Grèce
Turquie
Portugal
Syrie
Mer Méditerranée
Maroc
Algérie
Libye
Égypte