Introduction to code_aster

Warning:

In this document one describes the philosophy and the scopes of application of code_aster, without developing in detail the features available.

This document is a first making of contact with code_aster and was thus written with a preoccupation with a concision. It does not have the role to index all modelings or possible types of analysis with code_aster, and does not replace the plate of presentation which draws up a more exhaustive panorama of it.

All the information, provided here or in the various handbooks, is given to describe, with the maximum of precision, the contents of code_aster. They do not have as an ambition to deliver a formation with the digital modeling of the behavior of the mechanical structures. code_aster is only the establishment of methods described and shown in various works to which the engineer will have to refer, in complement of the reference material, if necessary. Handbooks of code_aster suppose acquired in addition a formation with the mechanics of the solids and the finite element method.

Contents

1 The study of the mechanical behavior of the structures

1.1 A code general practitioner

code_aster is a code general practitioner for the study of the mechanical behavior of the structures.

The priority scope of application is that of the mechanics of the deformable solids: that justifies the number of features attached to the mechanical phenomenon. However, the study of the mechanical behavior of the industrial components requires beforehand the modeling of the requests to which they are subjected, or of the physical phenomena which modify the parameters of this behavior (fluid external intern or, temperature, metallurgical phase shift, efforts of electromagnetic origin…). For these reasons, code_aster offer several possibilities of “chaining” of the mechanical phenomenon with the thermics or acoustics, or with of other software, as well as a “kit” of construction of problems thermo-hydro-mechanics coupled.

Although code_aster can be used for many problems of structural analysis (code general practitioner), it was developed in particular to allow the study of the components of materials or machines used in the field of the electricity production and transmission. Thus priority was given to the modeling of the metal structures, géomatériaux and the components of structure out of reinforced concrete. One will find very few modelings available for composite materials or elastomers.

The nonlinear analyses, as well in mechanics as in thermics, are in the middle of code_aster : their effective drug required the clarification of powerful and relatively simple algorithms of use, even if the goal is not to make them function in “black box”. For the complex studies, it is thus necessary to understand the nature of the operations carried out by the code, in order to be able to control them in an optimal way: one refers then to the theoretical notes giving the details of modelings and the methods, gathered in Manuel de Référence.

1.2 Method for calculation with code_aster

A structural analysis carried out with code_aster consist of the sequence of a certain number of orders described within a “command file” in format text. The engine and the interpreter of this command file are the language script PYTHON. It is thus possible to use all the features brought by PYTHON. See in particular documentations [U1.03.01], [U1.03.02] and examples of use [U1.05.00] and [U1.05.01]. Each order (for example reading of the grid, affection of the data material, linear static calculation) produced a “concept result”, which defines a structure of data that the user can handle and re-use in the further orders of calculation.

The syntax of all the orders is described and commented on in the U4 handbooks and U7 of the documentation of Use.

In order to simplify the task of the user, there exist total orders which gather the suitable sequence of operations for a certain number of calculation cases (for example static linear - order `MECA_STATIQUE`, nonlinear statics - order `STAT_NON_LINE`, nonlinear thermics - order `THER_NON_LINE`, etc). Some were developed directly in an integrated way, others are macros-orders in Python which manage the calls to various unit orders (like `ASSEMBLY` who allows to calculate and assemble the matrices of mass, damping and rigidity of a structure).

There exist also macro-orders especially dedicated to certain applications.

At the conclusion of a calculation, it is often possible to enrich the data-processing object containing the “concept got result”, by carrying out other calculations a posteriori: for example, starting from the field of displacements and of the constraints at the points of Gauss obtained in a mechanical calculation, one can calculate the field of deformations, the stress field interpolated to the nodes, etc.

These operations of postprocessing are carried out by the operators `CALC_CHAMP`, `POST_CHAMP`, `POST_ELEM`, `MACR_LIGNE_COUPE`, `POST_RELEVE_T`, etc.

1.3 Phenomena, modelings, finite elements and behaviors

1.3.1 Concepts

In code_aster, it PHENOMENON (order AFFE_MODELE) is a family of physical problems which use unknown factors and similar conservation equations: for example, the mechanical phenomenon makes call with the unknown factors of displacement, the thermal phenomenon with the unknown factors of temperature. According to modeling used, the number of unknown factors of this type can vary (for example one needs in each node only for one unknown factor for temperature in 3D, but one uses 3 unknown factors for the hulls).

In code_aster, MODELING (order AFFE_MODELE) described the manner according to which the continuous equations governing a given phenomenon are discretized, with the assistance of possible complementary assumptions (plane deformations, model of beam, model of hull…). In mechanics, for example, one can find plane modelings 3D, strains, plane stresses, hulls, plates, beams of Euler, beams of Timoshenko, pipes, etc… Each modeling uses a set of degrees of specific freedom: for example displacements in the three directions of space for modelings of continuous medium 3D, three displacements and three rotations for the hulls, etc.

The couple PHENOMENON/MODELING allows to assign in a bijective way a kind of finite element to each type of geometrical mesh.

In code_aster, one calls “finite element”, for a couple PHENOMENE/MODELISATION given, the unit consisted:

- The geometrical nature of the mesh support (representing a piece of volume or border: hexahedron, tetrahedron, triangle, quadrangle, segment…) ;
- The choice of the interpolations for the geometry and the unknown factors (functions of form);
- The “options” of calculation which the element can calculate (the operations for which the calculation of the adequate integrals was programmed: for example, elementary term of rigidity, elementary term of surface force, elementary term of mass…).
- Diagrams of digital integration (formulas of squaring of the Gauss type for example).

The behavior is at the base a physical notion related to the properties of material. It is expressed then in a mathematical way. For example, in mechanics, the relation of behavior is the relation which binds the stress field to the field of deformations. During a calculation, the relation of behavior is calculated in each point of integration (or not of Gauss).

code_aster fact the distinction enters the characteristics of material: orderS DEFI_MATERIAU and AFFE_MATERIAU and the relation of behavior (keyword BEHAVIOR).

1.3.2 Phenomena available in code_aster

code_aster is a software which makes mainly mechanics of the solids and structures but to supplement the representation of the environment of exploitation of the mechanical components, the choice was made include in code_aster features allowing the modeling of phenomena frequently associated with the mechanical phenomenon like thermics or acoustics.

1.3.2.1 The mechanical phenomenon

The mechanical phenomenon is modelled to achieve two main aims:

Firstly :
Determination of the internal state, in particular of the state of stress in each point of a structure, under various requests representing the conditions of operating. The knowledge of this state of stress makes it possible to continue the analysis of the mechanical behavior:

- to check the compliance with the rules of construction particular to each type of structure, in particular the Rules of Design or Construction (RCC…) ;
- to analyze the harmfulness of defects and their possible propagation: defects inherent in the development process of the component or the structure (geometrical inclusions, imperfections…) or resulting from the conditions of operating (cracking, erosion…) ;
- for the study of the behavior in cyclic loading and analysis with tiredness;
- for the prediction of the working loads with evolution of the internal state.

Secondly :

Determination of the deformed configuration induced by a permanent loading (static) or resulting from a slow evolution (quasi-static) or faster (dynamics) of the loadings or boundary conditions. The knowledge of this deformed configuration, and possibly speeds and corresponding accelerations, makes it possible to continue the analysis of the mechanical behavior:

- for the vibratory or acoustic behavior;
- for the transmission of the requests to other structures or components;
- for the analysis of the interactions with the close structures to determine the faulty operations or the parameters of wear which can result from it.

The levels of modeling which intervene for the study of this phenomenon are:

The representation of the structure starting from the geometrical form, with several possible modes of representation being able to coexist: • continuous medium corresponding to a geometry three-dimensionalE, or two-dimensional with various assumptions (forced plane, plane deformations, axisymetry complete or adapted to the decomposition of the loadings in modes of Fourier), • structural elements corresponding to a medium with average layer, a medium with average fibre or a discretized medium (beams, plates, hulls, pipes,…).
The representation of the behavior of materials, possibly different, in any point of a structure, with relations of behavior allowing to represent various conditions of use. Many relations of behavior are available: linear and non-linear elasticity, hyperelasticity non_linéaire, viscoelasticity, elastoplasticity, élasto-visco-plasticity, damage. The parameters of the relations of behavior can in general depend on variables known as “variable of orders” such as the temperature, the metallurgical state, the degree of hydration or drying of the concrete, the fluence (neutron irradiation), etc.
The representation of the boundary conditions and the loadings[1], for which one has features allowing to represent in any point of the structure, total reference mark or reference mark defined by the user: • Conditions of Dirichlet: imposed displacement, linear relations between components of displacement; • Conditions of Neumann: specific imposed force, surface and linear loadings, in particular allowing to represent the loadings of pressure; • Voluminal loadings, in particular allowing to represent gravity and the centrifugal loads of the bodies in rotation. • Conditions of unilateral, bilateral contact and of solid friction (Coulomb) These boundary conditions and loadings can depend on time (or of the frequency) and on one or more variables of space.

The non-linearities taken into account in the mechanical phenomenon are non-linearities of behavior, nonthe geometrical linearities (great displacements and great rotations, great deformations, buckling) and non-linearities of contact/friction.

1.3.2.2 The thermal phenomenon

It makes it possible to determine the thermal answer of solid media in permanent mode (stationary problem) or transient (evolutionary problem). One can model solid conduction, the convectif exchange, the heat transfer between walls, and the radiation ad infinitum. The thermal phenomenon can include modeling with the heating or the cooling of the metallurgical phase shift of steels, which makes it possible to simulate the operations of heat treatment or welding (the identification of the behavior is based on experimental diagrams TRC).

By analogy solved equations, the thermal phenomenon can also be used to model the hydration (the unknown factor is the degree of hydration) or the drying of the concrete (the unknown factor is the water concentration).

1.3.2.3 The acoustic phenomenon

The acoustic phenomenon is modelled to achieve two main aims:

- The study of the acoustic propagation in closed medium corresponding to the equation of Helmholtz in a compressible fluid, for fields of propagation to complex topology. The

1 A characteristic of code_aster is to assign the boundary conditions and the loadings of edges to meshs of edge which must be explicitly defined in the grid.

knowledge of the field of pressure makes it possible to continue the acoustic analysis to determine the field of noise levels (expressed in dB) and fields of active and reactive acoustic intensity.
- The study of the vibroacoustic coupled problems 3D corresponding to the vibratory behavior of a structure in a limited field of compressible, nonviscous fluid.

1.3.3 Phenomena coupled available in code_aster

So that there is no ambiguity, one will distinguish:
- The chaining of two phenomena (or weak coupling): prior study of the first phenomenon which one uses the results as data of the second;
- Complete coupling (or strong coupling) several phenomena: simultaneous resolution of the phenomena with actually coupled equations.

1.3.3.1 Internal chainings with code_aster

The chaining can be carried out inside code_aster or between this one and an external software.

Chainings currently carried out within code_aster are the following:
- Thermics – mechanics: all the mechanical characteristics of materials can depend on the temperature and the algorithms available for the mechanical phenomenon make it possible to exploit the results of a preliminary thermal calculation (unelastic deformations: thermal dilations, shrinking of the concrete…) carried out on a possibly different grid;
- Thermics – metallurgical: after a thermal calculation, it consists in calculating the proportions of the various metallurgical phases of steels;
- Thermics – metallurgical – mechanics: taking into account of four mechanical effects of the metallurgical transformations (deformation of phase shift, modification of the mechanical characteristics, plasticity of transformation, restoration of work hardening);
- Electric – mechanics: the electric coupling is limited to the taking into account of the forces of Laplace induced by currents of short-circuit in electric cables;
- Fluid – mechanics: assignment of field of pressure on a wall deduced from a calculation of mechanics of the fluids.

1.3.3.2 The couplings thermo-hydro-mechanics

The or not saturated saturated porous environments (géomatériaux, grounds, concrete) must be studied by coupling the three equations of mechanics, thermics and hydraulics. The user chooses the behaviors which it wishes to use among a kit of models thermo-hydro-mechanics known as `THM`. It can thus choose to take into account or not the effect of the temperature, and to represent one or two pressures. The choice of each behavior associated with the phenomena selected is carried out within this framework also.

1.3.3.3 Couplings for Interaction fluid-structure

Three types of couplings are available in the field of the interaction fluid-structure:
- The calculation of clean modes of a structure containing (or bathing in) a fluid at rest (with or without free surface);
- The calculation of the vibrations of a structure in a flow and the estimate of the damage while resulting by vibratory tiredness or wear;
- The taking into account of a boundary condition of type infinite fluid field.

1.4 Methods of analysis

To implement various modelings, one has several methods of analysis which correspond to various processes of application of the requests.

1.4.1 Static mechanics

Static analysis : it corresponds to permanent requests for the treatment of stationary thermics and the thermomechanical one. For the linear analyses, the got results can be combined linearly, according to the needs, and are usable to describe the initial state of an evolutionary process.

Quasi-static analysis : for all the mechanical processes where one can neglect the phenomena of inertia, implicit incrémentaux algorithms are available to solve the equations of non-linear behavior with taking into account of loadings and boundary conditions evolutionary.

1.4.2 Thermics

Stationary analysis : in linear thermics and not - linear, calculation of the thermal state of balance.
Transitory analysis : in linear and non-linear thermics, with possible taking into account of the metallurgical effects for metals and the hydration and drying for the concretes, like the problems of thermo-hydro-mechanics by neglecting the effects of inertia on the mechanical part.

1.4.3 Dynamic mechanics

For the processes where the effects of inertia and propagation must be taken into account, one speaks about dynamic analysis.
The analysis in physical base is the resolution of the equations in the classical base of the physical degrees of freedom.
The analysis in modal base consists of a preliminary calculation of the values and clean vectors of the structure, representing the "vibratory" state of the system. One project then equations to be solved on a basis of clean vectors: the number of degrees of freedom of the system to be solved is proportional in keeping with the modal base used.

For these two types of analyses in physical or modal base, the calculation of answer can be carried out into temporal or harmonic (in the linear case).

For the seismic analysis, one can also formulate the problem moving imposed in a relative reference frame (without the movement of training).

The linear dynamic analyses can be made by including the effects, of the second order on rigidity, the initial static stresses calculated as a preliminary (geometrical rigidity, centrifugal stiffening).

For the nonlinear problems, two methods of analysis are available:

- Analysis by modal recombination with boundary conditions nonlinear localised for problems with shock (operator `DYNA_VIBRA`);
- Nonlinear dynamic analysis in physical base (operator `DYNA_NON_LINE`).

1.4.4 Under-structuring

Under structuring consists in gathering several finite elements within a macronutrient and "to condense" the whole of their rigidity on the degrees of freedom (fewer) this macro - element.
The resolution of the total problem is limited then to the determination of the unknown factors carried by the macronutrients then with the calculation of the unknown factors carried by each "small" element in an independent way within each macronutrient.

The advantages of this method are the savings of time and memory, when the complete structure is made of reproduced elements several times by translation or rotation.

In dynamics, the modal analysis and calculation of the harmonic or transitory answer can be carried out in classical dynamic under-structuring by the methods of Craig-Bampton, Mac Neal or for the method known as of the modes of interface.

For the structures having a cyclic symmetry, the methods available make it possible to calculate the clean modes of the total structure starting from the dynamic behavior of a basic sector.

2 Method of resolution

For the resolution of the various mentioned problems, the principal method of space discretization currently established in code_aster is the finite element method.

2.1 A parameterized establishment of the finite element method

A special effort was tried to parameterize the establishment of the finite element method. The options of calculation necessary for each method of analysis (static, quasi-static, dynamic) and to each phenomenon (mechanical, thermal, acoustic) are treated overall for all the structure, whatever the modelings retained for a particular study.

Among the opportunities given by this architecture, let us quote:

- Independence enters the topology of discretization (`GRID`) and properties of interpolation of the finite elements assigned to these meshs (`MODEL`) from where the diversity of modelings usable on the same grid;
- The diversity of the relations of behavior and the properties of materials usable in the same model;
- Treatment of the boundary conditions and the loadings by specific finite elements of edge, to allow their localization without ambiguity, in particular for the continuous mediums;
- A systematic procedure allowing to treat the dependence of the material properties and the boundary conditions with various parameters (temperature, time, variable of space…) using `FUNCTION` or of `FORMULA` defined in Python.
- Structures of data allowing to use all modelings with the various algorithms of resolution.

2.2 A wide library of finite elements

The library of finite elements is parameterized to allow the assignment, with the various recognized meshs, of the discretized formulations of the phenomena available.

2.2.1 Continuous mediums

One calls continuous medium a portion of three-dimensional or two-dimensional structure, treated like a volume.

Modelings 3D are the simplest forms of continuous medium, because they do not call on any additional assumption. In modelings 2D, one removes an equation, but one must add assumptions: for example of plane strains or plane stresses in mechanics, axisymetry in thermics and mechanics.

2.2.2 Components of structure

The structural elements are built by integrating assumptions on the three-dimensional kinematic behavior (representing more or less well the phenomena of inflection, torsion, shearing, warping…). One can classify them in three categories:

1. Elements with average layer (plates, hulls): each type of element rests on assumptions of variation of the unknown factors in the thickness, which makes it possible to calculate the value in any point from that taken on the average layer (and possibly faces lower and higher in thermics);
2. Elements with average fibre (bars, beams, pipes, cables): the assumptions connect for each transverse section the value of the unknown factors in any point to that taken on average fibre;
3. Discrete elements (masses, springs, shock absorbers…): they make it possible to introduce on specific meshs or segments of the characteristics expressed into an unspecified reference mark.

These components require the use of the order `AFFE_CARA_ELEM` to define their characteristics.

2.2.3 Connections of modelings

The establishment retained for the finite element method makes it possible to treat structures modelled with various types of machine elements (continuous mediums or structural elements). The connection of finite elements being based on different degrees of freedom, in the same node, can be made by writing linear relations adapted to the nature of the connection. A particular methodology was developed to as correctly transmit as possible (within the meaning of least squares) the torques of effort. One can thus satisfactorily represent the connection between a medium 3D and beams, plates, hulls or pipes, as well as the connections hull-beam, hull-pipe or beam-pipe.

2.2.4 Discontinuous mediums

There exist also elements taking of account discontinuities (eg: crack)

1. By the method of the level-sets (elements XFEM) with modeling of the contact and friction;
2. By methods CZM (Cohesive Models Zone) with modeling of the refermeture contact and the damage in opening;
3. Elements with internal discontinuity;

2.2.5 Heterogeneous modelings

Techniques of homogenisation make it possible to represent with lower costs a network of tubes bathing in an incompressible fluid, multi-layer composite hulls, or beams multi - fibres.

3 Solveurs and linear algebra

Concerning the methods of classification of the unknown factors, of storage of the assembled matrices and resolution of the linear systems on which the various algorithms rest, one has several methods today: solvor direct or iterative, sequential or parallel.
In “native” in code_aster :

- Direct Solveurs: multi-frontal method (paralleled OpenMP) and method LDL^T.
- Iterative Solvor: prepacked combined gradient.

These methods are associated with algorithms of renumerotation of the degrees of freedom making it possible to optimize the size memory necessary to store the matrices.
Into external, it is possible of utiliSer following solveurs:

- MUMPS: solveurs direct paralleled out-of-core MPI;
- PeTSC: library of iterative solveurs, paralleled MPI;

For modal calculation: methods of Lanczos, Sorensen, the methods by opposite, for the real and complex problems, symmetrical power or not.

4 Tools of study

4.1 Complements and operations on the grid

Concept of grid used by code_aster is reduced very simple: list of the nodes and their coordinates, lists meshs and of their topology (connectivities). With these entities the notion of the groups of nodes and group of meshs is added. These groups make it possible to affect various characteristics of modeling (finite elements, materials, boundary conditions, loadings…) and to lead the examination of the results (selective extraction of components).

The user can create groups of nodes or meshs constantly in the course of calculation, thanks to logical or geometrical criteria. One can also modify the structure of data containing the grid: change of reference mark, addition of additional nodes on meshs, creation of new meshs or groups of meshs, destruction of meshs, etc the addition and the ablation of matter can be thus modelled simply.

The construction of a complex grid can be done only via the use of a tool more adapted to this use as integrated Salomé and its mailleurs (Netgen, BLSurf, Hexotic, etc) or GMSH.

4.2 Catalogue data material

A catalogue of data material under AQ gives access to the values of the parameters of laws of behavior for various materials usually used in the studies. The characteristic materials can be directly included in the command file thanks to a specific operator. For the free version, all the equipment of the catalogue is available but the base is empty.

4.3 Treatment and analysis of the results

4.3.1 Operations on the fields

The computed fields can be used in all kinds of algebraic combinations. In linear analysis, one can thus for example deduce the answer to a complex loading from the answers to the unit loadings on which it break up.

4.3.2 Statement of values

Operations of extraction of the fields of results are available on nodes or meshs. It is also possible to define a way of unspecified observation independent of the initial grid. Various calculations are proposed on the extracted fields (average, standard deviations, invariants tensorial, passage in local axes, etc). For the temporal or frequential evolutions, it is possible to extract the deformation at one moment (a frequency) or the answer from a particular size.

4.3.3 Impression of the results

The results can be printed in an easily consultable form or with the format of the tools for visualization (MED, I-DEAS, GMSH). code_aster communicate with Salomé using format MED.

One can also plot curves with various formats (postscript or other formats of images) using the tracer xmgrace.

4.4 Quality control of the results

Many features make it possible to control the quality of the results of a study or to facilitate its implementation of it.

4.4.1 Estimators of error and adaptive grid

Two categories of estimator of error are available. Coupled with the software of refinement déraffinement LOBSTER (chaining interns with code_aster via macros-order), they make it possible to adapt the grid in the course of calculation in order to reach a given precision, at a cost optimal calculation.

4.4.2 Checking of the quality of a modal base

Criteria of checking of the quality of a modal base make it possible to make sure that the number of clean modes selected makes it possible to correctly represent the phenomena which one wishes to study.

4.4.3 Use of incompatible grids

Operators of projection allow to continue on a second grid a calculation carried out on a first grid. One can thus use different grids in thermics and mechanics (by including for example a block fissures in the structure only at the time of his analysis in exploitation, after having calculated on a simpler grid the residual stresses due to its manufacturing process).

4.4.4 Automatic Recutting of the step of time and piloting of the loading

In the event of nonconvergence of the total algorithm of resolution, the user can ask so that the code engage of him even a recutting of the steps of time in order to allow convergence. A mechanism of management by events is also available.

In addition, it is also possible, in order to facilitate the convergence of calculations, to control the phased introduction of the loading by the value of a degree of freedom or a deformation (methods of continuation).

4.4.5 Indicators of discharge and loss of radiality

These indicators make it possible a posteriori to check the validity of the assumptions formulated on the nonlinear behavior of a structure, and the relevance of the mode of application of the loading selected (not of load).

5 Tool-dedicated

5.1 Definition and procedure

One calls tool-dedicated of the tools very related to the trade of owner of materials of production and electric distribution, and using code_aster as solvor. Tool-dedicated can have a more or less strong integration with code_aster. Two cases are distinguished:

- Integration with the command file code_aster as an macro-order (including the creation of the grid starting from simple geometrical data and call to an external maillor);
- Production by a separate tool (pre-post autonomous processor) of command files controlling calculation code_aster, and treatment in this tool of the files of the recovered results.

5.2 Modes of exchanges

code_aster can receive in data of the files coming from calculations carried out beforehand by external software. It can also export its results under an exploitable format by other tools. For certain types of analyses (for example interaction ground-structure or ground-fluid-structure with software MISS3D) the two types of chaining can be activated.

The exchanges with other software are currently done either with format MED, or in a format specific to the chained software. Several orders of code_aster the reading or the writing of the objects to be transmitted allows (fields of results, matrices, loadings…). In certain cases (MISS3D), macro-orders facilitate the implementation of a chained calculation.

5.3 Software interfaced with code_aster

Software of grid interfaced with code_aster are salome_meca, Gibi (subset of CASTEM2000), I - DEAS or GMSH. For the visualization of the results, one can use Salome_meca, I-DEAS or GMSH.