A simple example of use

Summary:

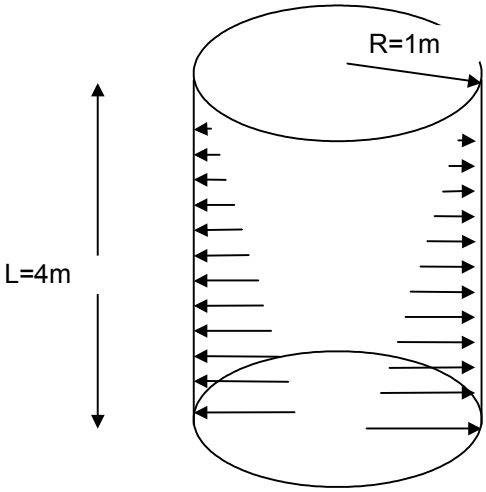
This document describes a very simple example of use of *Code_Aste*r. The files of setting in data are those of the case test FORMA00, available in the base of case tests of Code_Aster:

Script Salomé for the generation of the CAO and the grid	forma00.datg
Command file of Code_Aster	formed00.comm

On the basis of this simple calculation of a thin tank under hydrostatic pressure modelled into axisymmetric, one comments on the "essential" orders.

1 To model a mechanical problem with *Code_Aster*

The problem to be modelled is a thin cylindrical reserve constant thickness (thickness $0.1\,m$, interior ray $R=1\,m$, height $L=4\,m$) subjected to a pressure interns variable with the height, corresponding to a hydrostatic pressure.



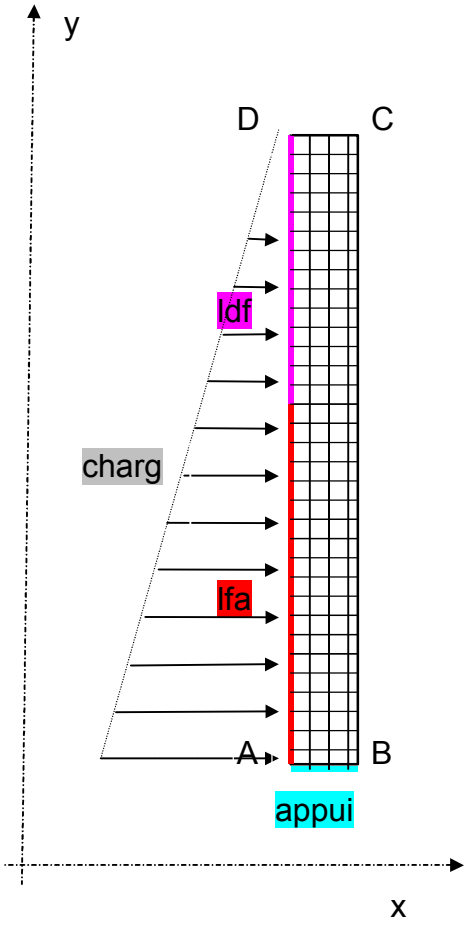
Being given symmetries of the geometry and loading, an axisymmetric two-dimensional modeling is chosen.

It will thus be enough to represent a vertical slice of this cylinder (in the plan XY)

The two stages to be envisaged are:

- the creation of the grid
- the drafting of the command file

2 Manufacturing of the grid: what to envisage?



Some is the software of grid used, it is necessary to envisage, as of the creation of the grid, to name the topological zones which will be used in calculation to affect elementary characteristics, boundary conditions, loadings, materials…

Indeed, although it is possible to directly use the numbers of nodes and meshs in the command file, it is preferable to use named entities (groups of nodes, groups of meshs). This makes it possible to have a command file independent of the degree of refinement of the grid, and a possible renumerotation of the nodes or elements.

In practice, these entities are groups:

- groups of nodes (container possibly only one node, like the points A, B, C, D in the example),
- groups of meshs corresponding to under - fields of the grid, or many meshs used to apply the loadings: here for example, the group of meshs `LDA` contains linear meshs (meshs of skin) which will be used to apply the pressure.

The simple script of generation of the geometry (a rectangle) and grid can be read in the file forma00.datg.

3 How to write its command file?

3.1 To start from nothing?

When one wants to model a new thermomechanical problem, one does not leave the blank sheet in general: it is useful to take as a starting point a command file of a modeling close to that to treat. How to obtain these files? The sources are varied:

- the base of the tests of *Code_Aster*, with its documentation, is often an important help, because it covers most of the features of the code (one can find these tests in the repertoire `astest` installation of the code),
- the formations make it possible to know thoroughly the whole of the orders referring to types of modelings: statics linear, thermal, dynamic, thermo - plasticity, postprocessing… In particular, practical works of the formations are tests: `FORMED ***`. Associated documentations of validation (V) contain intutitulés and corrected these practical works.

The drafting of this command file will be largely facilitated by using the editor of command file EFICAS.

3.2 Some essential orders

We now will detail the orders necessary to the realization of calculation considered.

Command file	Explanations
# Rolls thin under hydrostatic pressure	The comments are preceded by sign #,
`BEGINNING ();`	Obligatory order to start.
`e-mail = LIRE_MAILLAGE (FORMAT=' MED');`	Reading of the grid to format MED in the file associated by default with the grid: the logical unit 20. Creation of the concept `e-mail` containing the grid with the format Aster
# Definition of the model	
`modl=AFFE_MODELE (MAILLAGE=mail, AFFE=_F (TOUT=' OUI', PHENOMENE=' MECANIQUE', MODELISATION=' AXIS',),);`	A model is a concept, here of name `modl` , containing the types of finite elements useful for calculation. One associates with all the meshs of the grid of the axisymmetric mechanical finite elements. To note: the same order can continue on several lines
# Definition of material	
`acier=DEFI_MATERIAU (ELAS=_F (E=210000000000.0, NU=0.3,),);`	Definition of a particular material, which one chose to name here `steel`, and of its characteristics: The Young modulus and the Poisson's ratio in the case of an elastic material.
# Assignment of material on the grid	

```
chmat=AFFE_MATERIAU (MAILLAGE=mail,
                     AFFE=_F (TOUT=' OUI',
                              MATER=acier,),);
```

Assignment of material `steel` on the grid `e-mail`.
Here the material is the same one for all the grid
One could of course affect different materials on particular groups of meshs.

Definition of the boundary conditions

```
climatisation=AFFE_CHAR_MECA (MODELE=modl,
                  FACE_IMPO=_F (GROUP_MA=' LAB',
                                DY=0,),);
```

The boundary conditions can relate to nodes, groups of nodes, meshs or groups of meshs.

Here nodes of the group of meshs `LAB` (meshs of edge) are affected following condition:

$DY=0$ what means: "following displacement y " no one for all the nodes of the group `LAB`

Definition and assignment of the loading: pressure function of y

```
f_y=DEFI_FONCTION (NOM_PARA=' Y',
                   VALE= (0.0, 200000.0,
                          4.0,0.0,),);
```

The functions are point by point defined (variation closely connected between two points by default)
Here, the pressure varies between:
$200000\,Pa$ for $y=0$
and 0 for $y=L$

```
charg=AFFE_CHAR_MECA_F (MODELE=modl,
             PRES_REP=
                        _F (GROUP_MA= ('LFA', 'LDF',),
                            PRES=f_y,),);
```

Assignment of the pressure (function of y) on edge made up of the groups of meshs `LFA` and `LDF`

Resolution

```
res1=MECA_STATIQUE (MODELE=modl,
                    CHAM_MATER=chmat,
                                  EXCIT=  (_F
(CHARGE=charg,),
                                           _F
(CHARGE=climatisation,),),);
```

Total ordering of resolution of the static problems in linear thermoelasticity

Association with the model, the field of material (X) and it (S) loading (S) previously definite.

`res1` is the name of the concept result produced by the order. It contains in particular the field of calculated displacement.

Calculation of the constraints

```
res1=CALC_CHAMP (reuse =res1,
                 RESULTAT=res1,
                 CONTRAINTE=' SIGM_ELNO',);
```

`reuse=res1` mean that one "enriches" the concept.

`res1` : the stress field will be stored besides the field of displacements already present in the concept.

The name `'SIGM_ELNO'` mean "constraints calculated with the nodes of each element starting from displacements"

Impression of the results

```
IMPR_RESU (FORMAT=' MED',
           RESU=_F (RESULTAT=res1,),);
```

Impression of the results to format MED: displacements/forced on all the grid

`IMPR_RESU (RESU=_F (RESULTAT=res1, GROUP_NO=' A',),);`	Impression of the results to the format text for the only group of nodes "with".
`END ();`	Obligatory order to close an execution

4 What contains the file of results?

A heading pointing out the date, the version of the code, the data-processing platform of execution used:

```
        -- CODE_ASTER -- VERSION: DEVELOPMENT STABILISEE (testing) --

                        Version 11.1.0 of the 12/7/2011
                      Copyright EDF R & D 1991 - 2012
                  Execution of: Thu Apr 12 18:41: 37 2012
                        Name of the machine: cli75at
                            Architecture: 64bit
                        Type of processor: x86_64
                Operating system: Linux 2.6.32-27-generic
                       Language of the messages: Fr (UTF-8)

                         Parallelism MPI: inactive
                    Version of bookstore HDF5: 1.8.4
                    Version of bookstore MED: 3.0.4
                        Bookstore MUMPS: installed
                   Version of the bookstore SCOTCH TAPE: 5.1.10
                   Limit of the static storage: 1,000 Mo
                 Limit of the dynamic storage: 299,000 Mo
               Size limits files of exchange: 48,000 Go


 --------------------------------------------------------------------------------
 ASTER 11.01.00 CONCEPT res1 CALCULATES 4/12/2012 A 18:41: 37 OF TYPE EVOL_ELAS
```

Impression of the field of displacements to the nodes of the group A (in fact only one node contains: the point A) :

```
GROUP_MA: NOEUDA
FIELD WITH THE NODES OF REFERENCE SYMBOL  DEPL
 SEQUENCE NUMBER: 1 INST:  0.00000E+00
 NODE        DX           DY
 N1        9.95605E-06  1.24077E-23
```

Impression of the stress field by elements to the nodes:

```
 FIELD BY ELEMENT WITH THE NODES OF REFERENCE SYMBOL  SIGM_ELNO
 SEQUENCE NUMBER: 1 INST:  0.00000E+00
 M211      SIXX         SIYY         SIZZ         SIXY
 N1       -1.26603E+05 -8.29982E+04  1.97655E+06  1.02990E+03
 M212      SIXX         SIYY         SIZZ         SIXY
 N1       -1.50710E+05 -1.11440E+04  1.94959E+06 -2.92821E+04
```

A table summarizing the orders used and time CPU of each one:

```
 ********************************************************************************
 * COMMAND                 :       TO USE:     SYSTEM:   USER+SYS:     ELAPSED *
 ********************************************************************************
 * AFFE_MODELE             :       0.08:       0.02:       0.10:       0.10 *
 ********************************************************************************
 * TOTAL_JOB               :       1.59:       0.14:       1.73:       1.67 *
 ********************************************************************************
```

5 And other files produced by calculation?

5.1 The file of messages

This file contains the echo of the orders and gives additional information about the execution of each order:

for example MECA_STATIQUE :

```
#------------------------------------------------------------------------
# Orders No:  0009            Concept of the type: evol_elas
#------------------------------------------------------------------------
  res1=MECA_STATIQUE (EXCIT= (_F (TYPE_CHARGE=' FIXE',
                                  CHARGE=charg),
                             _F (TYPE_CHARGE=' FIXE',
                                 CHARGE=climatisation)),
                      INFO=1,
                      OPTION=' SIEF_ELGA',
                      SOLVEUR=_F (RENUM=' METIS',
                                  STOP_SINGULIER=' OUI',
                                  METHODE=' MULT_FRONT',
                                  NPREC=8),
                      INST=0.0,
                      MODELE=modl,
                      CHAM_MATER=chmat,);

The linear system to solve contains 51 nodes of which:
   - 43 nodes carrying of the physical degrees of freedom
   - 8 nodes carrying of the degrees of freedom of Lagrange
For a total of 94 equations.

   FIELD STORES:             URGENT DEPL:  0.00000E+00  SEQUENCE NUMBER:     1

# End orders No: 0009 user+syst: 0.13s (syst: 0.02s, Elaps: 0.10s)
#------------------------------------------------------------------------
```

5.2 The file of results to format MED

The file of results to format MED is produced by default by Astk in the logical unit 80.

This file MED can be been essential by the modules of visualization of Salomé (POSTPRO, PARAVIS) in order to display the fields which compose it (displacement, constraints):