A simple example of Résumé

use:

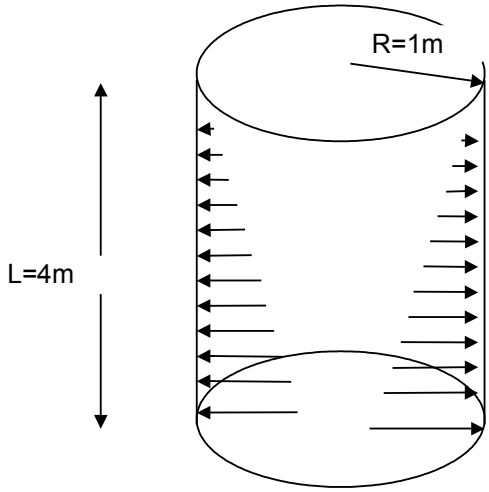
This document describes a very simple example of use of *Code_Aster*. The files of setting in data are those of the case test DEMO005, available in the base of case tests of Code_Aster:

Salomé scripting for the generation of the CAO and the mesh	demo005a.datg
Fichier of commands of Code_Aster	demo005a.com

Sur the base of this simple computation of a thin tank under hydrostatic pressure modelized into axisymmetric, one comments on the "essential" commands.

1 To modelize a mechanical problem with *Code_Aster*

the problem with modelizing is a thin cylindrical reserve of constant thickness (thickness $0.1\,m$, interior radius $R=1\,m$, height $L=4\,m$) subjected to a variable internal pressure with the height, corresponding to a hydrostatic pressure.



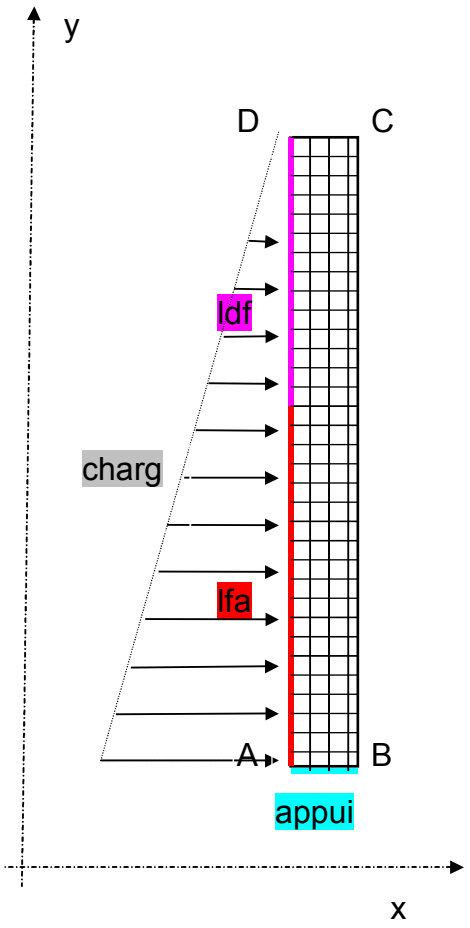
Being given symmetries of the geometry and loading, an axisymmetric two-dimensional modelization is chosen.

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

Les two stages to be envisaged are:

the creation of the mesh
the drafting of the command file

2 Fabrication of the mesh: what envisage?



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

En effet, although it is possible to directly use the numbers of nodes and meshes in the command file, it is preferable to use named entities (nodes groups, mesh groups). This makes it possible to have a command file independent of the degree of refinement of the mesh, and a possible renumbering of the nodes or elements.

In practice, these entities are groups:

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

The simple scripting of generation of the geometry (a rectangle) and mesh can be read in the file demo005a.datg.

3 How to write its command file?

3.1 Start from nothing?

When one wants to modelize 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 modelization close to that with treating. How to obtain these files? The sources are varied:

the base of the tests of *Code_Aster*, with its documentation, is often an important aid, because it covers most of the functionalities of the code (one can find these tests in the directory `astest` of the instalment of the code),
trainings make it possible to know thoroughly all the commands referring to types of modelizations: statics linear, thermal, dynamic, thermo - plasticity, postprocessing… En particulier, hands-on training of trainings are tests: `FORMA ***`. 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 commands

Nous now let us detail the commands necessary to the realization of computation considered.

Command file	thin
Explications # Cylindre under hydrostatic pressure	Les comments are preceded by sign #,
`DEBUT ();`	Compulsory command to start.
`e-mail = LIRE_MAILLAGE (FORMAT='MED' );`	Reading of the mesh to format MED in the file associated by defect with the mesh: the logical unit 20. Creation of the concept `e-mail` containing the mesh with the format Aster
# Définition 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 computation. One associates with all the axisymmetric meshes of the element mesh finished mechanical. To note: the same command can continue on several lines
# Définition of the material	
`acier=DEFI_MATERIAU (ELAS=_F (E=210000000000.0, NU=0.3,),);`	Definition of a particular material, that 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 the material on the mesh	

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

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

Definition of the boundary conditions

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

The boundary conditions can relate to nodes, nodes groups, meshes or mesh groups.

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

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

Définition 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)
Ici, 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 mesh groups `LFA` and `LDF`

Résolution

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

Total command of resolution of the static problems in linear thermoelasticity

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

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

Computation of the stresses

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

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

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

Name `"SIGM_ELNO"` means "forced calculated with the nodes of each element starting from displacements"

Impression of results

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

Printing of the results to format MED: displacements/stresses on all mesh

```
IMPR_RESU (RESU=_F (RESULTAT=res1,
                    GROUP_NO='A',),);

FIN ();
```

Printing of the results to the format text for only nodes group "A".

Does compulsory command to close a Que

4 execution contain the results file?

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

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

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

                         Parallélisme MPI: idle
                    Version of bookshop HDF5: 1.8.4
                    Version of bookshop MED: 3.0.4
                        Bookshop MUMPS: installed
                    Version of bookshop SCOTCH: 5.1.10
                    Limite of the static storage: 1,000 Mo
                  Limite of the dynamic storage: 299,000 Mo
                Limiting Taille of the files of exchange: 48,000 Go


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

Printing 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 SYMBOLIC NAME  DEPL
 SEQUENCE NUMBER: 1 INST:  0.00000E+00
 NODE        DX           DY
 N1        9.95605E-06  1.24077E-23
```

Printing of the stress field by elements to the nodes:

```
 FIELD PAR ELEMENT WITH THE NODES OF SYMBOLIC NAME  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 commands used and time CPU of each one:

```
 ********************************************************************************
 * COMMAND                 :       USER:     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 computation?

5.1 The file of messages

This file contains the echo of the commands and gives additional informations on the execution of each command:

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=clim)),
                      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
Pour a total of 94 equations.

   FIELD STORES:             DEPL TIME:  0.00000E+00  Sequence number:     1

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

5.2 The results file with format MED

the results file with format MED is produced by defect by Astk in the logical unit 80.

This file MED can been essential by the moduli of visualization of Salome (POSTPRO, PARAVIS) in order to display the fields which compose it (displacement, stresses):