
Title: *Simple operating example*
Author(s): **M. ABBAS, J.M. PROIX, N. TARDIEU**
Translator(s): **C. LUZZATO**

Date : 26/05/05
Key: U1.05.00-C Page : 1/10

Organisation(s) : EDF-R&D/AMA

User Manual
Booklet U1.0- : Introduction to Code_Aster
Document : U1.05.00

Simple operating example

Summary:

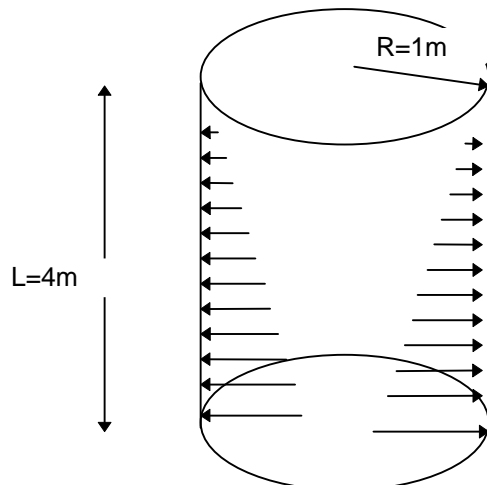
This document describes a very simple example of the operating of *Code_Aster*.

We show all of the « fundamental » commands through a reservoir calculation (thin cylinder subjected to hydrostatic pressure) modelled with axis-symmetry.

The command and result files are both analysed.

1 Modelling a mechanical problem with Code_Aster

The problem that needs to be modelled consists of a thin cylindrical reservoir (0.02M thickness, 1m, height $L=4\text{m}$) subjected to an internal pressure which varies with height: this corresponds to a hydrostatic pressure.



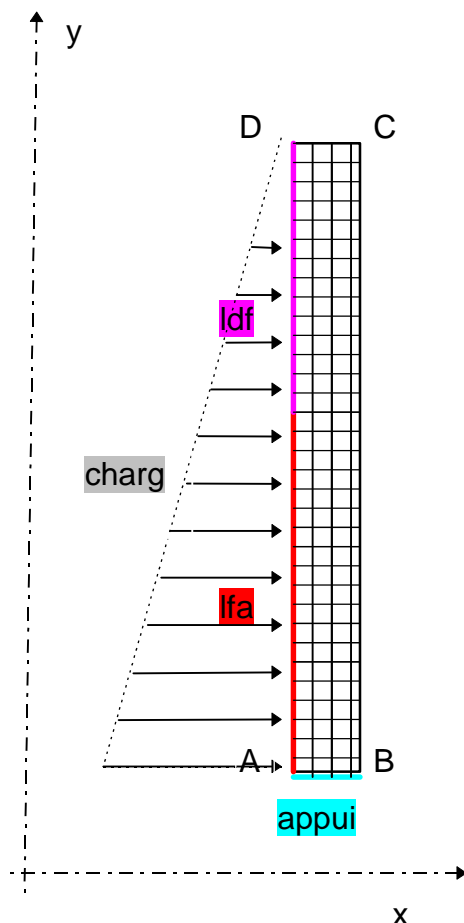
Considering that the cylinder and its loading are symmetrical, we can model the problem by using two dimensional axis-symmetry.

We only need to represent a vertical slice of the cylinder (in its XY plane).

The two steps to consider are

- The creation of the mesh
- The writing of the command file

2 Creating the mesh: what should be expected



Whatever the mesh generation software used (GIBI, IDEAS, GMSH), we must name all of the entities which will be used during the calculation for the assigning of elementary characteristics, boundary conditions, loadings, materials...

Even though we could directly use the node and mesh element numbers in the command file, it is easier to refer to named entities. This allows us to have a command file which is independent from the degree of refinement of the mesh, and thus unaffected by a re-numbering of nodes or elements.

In practice, these entities are groups :

- Node groups (which can eventually contain only one node, such as the A, B, C, D points from the example)
- Mesh element groups, which correspond to sub-domains of the mesh, or to mesh elements used to apply a load. Here for example, the `lfa`, `ldf` mesh groups contain lineic mesh elements (skin mesh elements) which will be use to set the precision.

3 How to write the command file

3.1 Starting from scratch

When we want to model a new Thermo-mechanic problem, we usually start with a clean slate. However, it can be helpful to use command files for similar models as inspiration. How can we obtain these files ? There are several sources:

- The *Code_Aster* test database and documentation. It can often be of great help as it covers an important part of the functionalities of the code (we can find these tests in the *astest* folder),
- The courses; they are designed so that the user can thoroughly master all of the commands relating to modelling types: linear static, thermal, dynamic, thermo-plasticity, post-processing...

The EFICAS command file editor greatly simplifies the writing of the command file.

3.2 Analysis of commands

We are now going to detail the commands that are necessary to the realisation of the proposed calculation.

Command file	Analysis
<pre># Title thin cylinder subjected to hydrostatic pressure DEBUT () ; PRE_GMSH() ; mail = LIRE_MALLAGE () ; # Redefining of node groups and mesh element groups mail=DEFI_GROUP(reuse =mail, MAILLAGE=mail, CREA_GROUP_MA=(_F(NOM='APPUI', GROUP_MA='GM11',), _F(NOM='LDF', GROUP_MA='GM13',), _F(NOM='LFA', GROUP_MA='GM14',), _F(NOM='ND_A', GROUP_MA='GM1',), _F(NOM='ND_B', GROUP_MA='GM2',), _F(NOM='ND_C', GROUP_MA='GM3',), _F(NOM='ND_D', GROUP_MA='GM4',),),), # Defining of model</pre>	<p>Comments are preceded by the # sign</p> <p>Mandatory starting command... The mesh is in the GMSH format Reading of the mesh from the mesh file, and creation of the <code>mail</code> concept which contains the mesh in the Aster format.</p> <p>Definition of the mesh element groups from the ones created in GMSH</p> <p>A Model is a concept which contains the finite element types which will be useful for the calculations</p>

Title: *Simple operating example*
Author(s): **M. ABBAS, J.M. PROIX, N. TARDIEU**
Translator(s): **C. LUZZATO**

Date : 26/05/05
Key: U1.05.00-C Page : 4/10

```
mod1=AFFE_MODELE(MAILLAGE=mail,  
                AFFE=_F(TOUT='OUI',  
                        PHENOMENE='MECANIQUE',  
                        MODELISATION='AXIS',),),);  
  
# Defining of material  
  
acier=DEFI_MATERIAU(ELAS=_F(E=210000000000.0,  
                             NU=0.3),),);  
  
chmat=AFFE_MATERIAU(MAILLAGE=mail,  
                   AFFE=_F(TOUT='OUI',  
                           MATER=acier),),);  
  
# Definition of boundary conditions  
  
clim=AFFE_CHAR_MECA(MODELE=mod1,  
                   FACE_IMPO=_F(GROUP_MA='APPUI',  
                                 DY=0),),);  
  
# Definition of load : pressure as a function  
of Y  
  
f_y=DEFI_FONCTION(NOM_PARA='Y',  
                 VALE=(0.0,20000.0,  
                      4.0,0.0),),);  
  
charg=AFFE_CHAR_MECA_F(MODELE=mod1,  
                      PRES_REP=  
                        _F(GROUP_MA=('LFA','LDF'),  
                          PRES=f_y),),);  
  
# Resolution  
  
res1=MECA_STATIQUE(MODELE=mod1,  
                  CHAM_MATER=chmat,  
                  EXCIT=( _F(CHARGE=charg),  
                          _F(CHARGE=clim),),),);
```

Associates axis-symmetrical mechanical finite elements to all the mesh elements.

One single command can be continued over several lines.

The characteristics of each material which constitutes the mesh is inputted
Young modulus and Poisson coefficient.

The material is assigned to the mesh
Note that here the material is the same for all of the mesh.
Otherwise, different types of materials could be assigned to different mesh element groups.

The boundary conditions can concern nodes, mesh elements or mesh element groups.

The nodes from the APPUI mesh element group (edge mesh elements) are assigned the condition :
DY = 0 which means:
No « displacement along the y axis»

The functions are defined point by point (linear variation between two points is the set default)

Here the pressure varies between :
20000 Pa for y=0
and 0 for y = L

The pressure (depending on y) is assigned to the edge containing the mesh element group
LFA LDF

Global command for the resolution of static problems in linear thermo-elasticity

The material field
We define the loads

res1 is the name of the concept result containing the displacement field

Title: *Simple operating example*
Author(s): **M. ABBAS, J.M. PROIX, N. TARDIEU**
Translator(s): **C. LUZZATO**

Date: 26/05/05
Key: U1.05.00-C Page: 5/10

```
# constraint calculation

res1=CALC_ELEM(reuse =res1,

                RESULTAT=res1,
                OPTION='SIGM_ELNO_DEPL',);

# Printing of the results for points A B C D

IMPR_RESU(MODELE=mod1,
          RESU=_F(RESULTAT=res1,
                 GROUP_MA='ND_A',),),);

IMPR_RESU(MODELE=mod1,
          RESU=_F(RESULTAT=res1,
                 GROUP_MA='ND_B',),),);

IMPR_RESU(MODELE=mod1,
          RESU=_F(RESULTAT=res1,
                 GROUP_MA='ND_C',),),);

IMPR_RESU(MODELE=mod1,
          RESU=_F(RESULTAT=res1,
                 GROUP_MA='ND_D',),),);

# Printing of results
IMPR_RESU(MODELE=mod1,
          RESU=_F(RESULTAT=res1,),),);

# Printing of results with GMSH visualisation

DEFI_FICHIER ( ACTION='ASSOCIER',
              UNITE=37,)

IMPR_RESU( MODELE=mod1,
          FORMAT='GMSH',
          UNITE=37,
          RESU=_F(RESULTAT = res1,),)

DEFI_FICHIER ( ACTION='LIBERER',
              UNITE=37,)

FIN();
```

reuse=res1 means that we « enrich » the concept
res1 : the constraint field will be stored in addition to the displacement field.

The name 'SIGM_ELNO_DEPL' means "constraint calculated at the node of each element from the displacements"

Printing results in text format
Displacement at the nodes corresponding to points A B C D

Printing results in text format
Displacement/constraints for the whole mesh

Defining the logical unit for the GMSH file

Printing results in the GMSH

Closing the logical unit

...
Mandatory command to finish an execution

Title: *Simple operating example*
Author(s): **M. ABBAS, J.M. PROIX, N. TARDIEU**
Translator(s): **C. LUZZATO**

Date : 26/05/05
Key: U1.05.00-C Page : 6/10

4 What does the result file contain

=> A header which includes the date, the version and the platform used:

```
-- CODE_ASTER -- VERSION DE DEVELOPPEMENT 7.04.00 --  
  
COPYRIGHT EDF-R&D 2003  
  
EXECUTION DU : ME-11-JUIN-2003 10:11:32  
  
PLATE-FORME : CLALASTR.CLA.EDF  
  
NB MAX PROC : 1  
  
SYSTEME : OSF1  
  
CPU : ALPHA
```

ASTER 7.01.07 CONCEPT res1 CALCULE LE 11/06/2003 A 10:11:34 DE TYPE EVOL_ELAS

=> Printing the displacement field at the nodes of the A,B,C,D groups (points)

```
GROUP_MA : ND_A  
CHAMP AUX NOEUDS DE NOM SYMBOLIQUE DEPL  
NUMERO D'ORDRE: 1 INST: 0.00000E+00  
NOEUD DX DY  
N1 4.68143E-06 3.74958E-24
```

```
GROUP_MA : ND_B  
CHAMP AUX NOEUDS DE NOM SYMBOLIQUE DEPL  
NUMERO D'ORDRE: 1 INST: 0.00000E+00  
NOEUD DX DY  
N2 4.65280E-06 5.20865E-24
```

```
GROUP_MA : ND_C  
CHAMP AUX NOEUDS DE NOM SYMBOLIQUE DEPL  
NUMERO D'ORDRE: 1 INST: 0.00000E+00  
NOEUD DX DY  
N3 1.20218E-06 -2.63963E-06
```

```
GROUP_MA : ND_D  
CHAMP AUX NOEUDS DE NOM SYMBOLIQUE DEPL  
NUMERO D'ORDRE: 1 INST: 0.00000E+00  
NOEUD DX DY  
N4 3.31016E-09 -2.81696E-06
```

=> Printing the constraint field

```
CHAMP PAR ELEMENT AUX NOEUDS DE NOM SYMBOLIQUE SIGM_ELNO_DEPL  
NUMERO D'ORDRE: 1 INST: 0.00000E+00  
M37 SIXX SIYY SIZZ SIXY  
N3 -5.13918E+03 -1.03624E+04 2.42856E+05 6.82275E+01 => Point C  
N54 -4.40750E+03 -9.63147E+03 2.44564E+05 -4.70082E+01  
N55 4.14124E+03 1.04408E+04 3.02923E+05 -6.66820E+01  
N24 3.26306E+03 9.56344E+03 3.00873E+05 4.85536E+01  
M51 SIXX SIYY SIZZ SIXY  
N11 -3.38160E+03 4.89698E+03 9.41453E+05 2.91041E+04 => Point B  
N68 -2.46101E+03 3.32521E+03 9.46093E+05 2.90567E+04  
N10 -9.31239E+02 9.59144E+03 9.65451E+05 -3.32524E+04  
N2 -1.90427E+03 1.11108E+04 9.60688E+05 -3.32050E+04  
M111 SIXX SIYY SIZZ SIXY
```

Title: *Simple operating example*
Author(s): **M. ABBAS, J.M. PROIX, N. TARDIEU**
Translator(s): **C. LUZZATO**

Date : 26/05/05
Key: U1.05.00-C Page : 7/10

N110	-2.00922E+04	-1.00398E+04	9.51623E+05	2.92456E+04	
N50	-1.90559E+04	-1.15088E+04	9.56546E+05	2.92049E+04	
N1	-1.61470E+04	-4.53210E+03	9.76893E+05	-3.34251E+04	=> Point A
N7	-1.72353E+04	-3.11525E+03	9.71848E+05	-3.33844E+04	
M112	SIXX	SIYY	SIZZ	SIXY	
N3	3.43073E+03	9.63409E+03	2.51426E+05	5.11140E+01	=> Point C

(belongs to several mesh elements)

N25	-4.96037E+03	-1.02898E+04	1.93404E+05	5.97663E+01	
N111	-4.37491E+03	-9.70470E+03	1.94770E+05	-5.55617E+01	
N54	4.16274E+03	1.03658E+04	2.53134E+05	-6.42140E+01	

=> A table summarising the employed commands and the CPU time used for each:

```
*****
*  COMMANDE          *      USER *      SYSTEME *      TOTAL *
*****
*  DEBUT             :      0.02 :      0.03 :      0.05 *
*  PRE_GMSH          :      0.02 :      0.02 :      0.03 *
*  LIRE_MAILLAGE     :      0.02 :      0.00 :      0.02 *
*  DEFI_GROUP        :      0.02 :      0.00 :      0.02 *
*  AFFE_MODELE       :      0.02 :      0.00 :      0.02 *
*  DEFI_MATERIAU     :      0.07 :      0.00 :      0.07 *
*  AFFE_MATERIAU     :      0.00 :      0.00 :      0.00 *
*  AFFE_CHAR_MECA    :      0.03 :      0.00 :      0.03 *
*  DEFI_FONCTION     :      0.00 :      0.00 :      0.00 *
*  AFFE_CHAR_MECA_F  :      0.02 :      0.02 :      0.03 *
*  MECA_STATIQUE     :      0.08 :      0.03 :      0.12 *
*  CALC_ELEM         :      0.03 :      0.00 :      0.03 *
*  IMPR_RESU         :      0.08 :      0.02 :      0.10 *
*  IMPR_RESU         :      0.03 :      0.02 :      0.05 *
*  IMPR_RESU         :      0.05 :      0.00 :      0.05 *
*  IMPR_RESU         :      0.05 :      0.00 :      0.05 *
*  IMPR_RESU         :      0.05 :      0.00 :      0.05 *
*  IMPR_RESU         :      0.17 :      0.20 :      0.37 *
*  FIN               :      0.00 :      0.03 :      0.03 *
*****
*  TOTAL_JOB         :      0.83 :      0.42 :      1.25 *
*****
```

5 And what about the other files created during the calculation

5.1 The MESSAGE file

This file contains all of the processed commands and gives further information about the execution of each command.

For example MECA_STATIQUE :

```
# -----
#  COMMANDE NO : 0011          CONCEPT DE TYPE : evol_elas
#  -----
res1=MECA_STATIQUE(CHAM_MATER=chmat,
                  MODELE=mod1,
                  ANGLE=0,
                  NIVE_COUCHE='MOY',
                  NUME_COUCHE=1,
                  SOLVEUR=_F(NPREC=8,
                             METHODE='MULT_FRONT',
                             STOP_SINGULIER='OUI',
                             RENUM='METIS'),
                  INFO=1,
                  PLAN='MAIL',
                  INST=0.0,
                  EXCIT=( _F(CHARGE=charg,
                             TYPE_CHARGE='FIXE'),
                          _F(CHARGE=clim,
                             TYPE_CHARGE='FIXE') ),
                  );

--- NOMBRE TOTAL DE NOEUDS : 138 DONT :
      12 NOEUDS "LAGRANGE"
--- NOMBRE TOTAL D'EQUATIONS : 264
--- NOMBRE DE COEFFICIENTS NON NULS DANS LA MATRICE : 2120
--- NOMBRE DE BLOCS UTILISES POUR LE STOCKAGE : 1
```

5.2 The GMSH file

The GMSH file (written with IMPR_RESU, 'GMSH' format) contains all of the data necessary to visualise the model.

The same thing is possible for all other interface files using post-processing graphical software (IDEAS, ENSIGHT)

Or the XMGRACE plotter.

5.3 Generating the mesh with GMSH

We describe here the `geo` file which allows generating the mesh with the GMSH software (Free and open source software).

```
////////////////////////////////////
// Reservoir meshing - GMSH V1.60
////////////////////////////////////

// Variables
// Reservoir internal radius
Rint = 1;
// Wall thickness
ep   = 0.02;
```

Title: *Simple operating example*
Author(s): **M. ABBAS, J.M. PROIX, N. TARDIEU**
Translator(s): **C. LUZZATO**

Key: *U1.05.00-C* Date: *26/05/05*
Page: *9/10*

```
// Reservoir external radius
Rext = Rint+ep;
// Reservoir height
h = 4;
// Number of elements on the horizontal axis
nbelh= 5;
// Nombre of elements on the vertical axis
nbelv= 20;

// Points
Point(1) = {Rint,0,0,1};
Point(2) = {Rext,0,0,1};
Point(3) = {Rext,0.75*h,0,1};
Point(4) = {Rext,h,0,1};
Point(5) = {Rint,h,0,1};
Point(6) = {Rint,0.75*h,0,1};

// Lines
Line(1) = {1,2};
Line(2) = {2,3};
Line(3) = {3,4};
Line(4) = {4,5};
Line(5) = {5,6};
Line(6) = {6,1};
Line(7) = {6,3};

// ruled surface for ruled mesh
Line Loop(1) = {-2,-1,-6,7};
Ruled Surface(1) = {1};
Line Loop(2) = {3, 4, 5, 7};
Ruled Surface(2) = {2};

// Description of the mesh elements/nodes (points) Physical Point(1) = {1};
Physical Point(2) = {2};
Physical Point(3) = {3};
Physical Point(4) = {4};
Physical Point(5) = {5};
Physical Point(6) = {6};

// Description of the mesh elements/nodes (lines)
Physical Line(11) = {1};
Physical Line(12) = {4};
Physical Line(13) = {5};
Physical Line(14) = {6};

// Description of the mesh elements/nodes (surface)
Physical Surface(21) = {1,2};

// mesh refinement (ruled)
Transfinite Line{1} = nbelh+1;
Transfinite Line{4} = nbelh+1;
Transfinite Line{7} = nbelh+1;
Transfinite Line{5,3}= 0.25*nbelv+1;
Transfinite Line{2,6}= 0.75*nbelv+1;

Transfinite Surface{1} = {3,6,1,2};
Transfinite Surface{2} = {3, 4, 5, 6};

// transformation from triangles to quadrangles
Recombine Surface {1,2};
```

GMSH will generate the mesh element groups 1 through 6 (points), 11 through 14 (lines) and the group 21 (Surface) The mesh element groups are retrieved *Code_Aster* with the name GM## where ## is the number of the GMSH group.

In our example, we redefine these names with `DEFI_GROUP` pour to obtain a more explicit name (APPUI, LFA, LDE).

Title: *Simple operating example*
Author(s): **M. ABBAS, J.M. PROIX, N. TARDIEU**
Translator(s): **C. LUZZATO**

Date : 26/05/05
Key: U1.05.00-C Page : 10/10

Page intentionally left blank.