

Title : *An operating example for Code_Aster : calculation of a bent pipe*
Author(s) : **M. ABBAS, J.M. PROIX**
Translator(s) : **C. LUZZATO**

Date : 18/05/05
key : U1.05.01-B Page : 1/10

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

User Manual
Booklet U1.0- : Introduction to Code_Aster
Document : U1.05.01

An operating example for Code_Aster : Calculation of a bent pipe

Summary :

This document describes a simple operating example for *Code_Aster*. It is included in the download procedure of *Code_Aster* available on the site code-aster.org.

1 Problem data

1.1 Geometry

The study concerns pipework which contains two strait pipes and a bend [Figure 1.1-a].

The Geometric data of the problem is the following:

- The length L_G of each pipe is 3 m,
- The radius R_C of the bend 0.6 m,
- The angle θ of the bend is 90 degrees,
- The thickness of the pipes and the bend is 0.02 m,
- And the internal radius R_e of the pipes and the bend is 0.2 m.

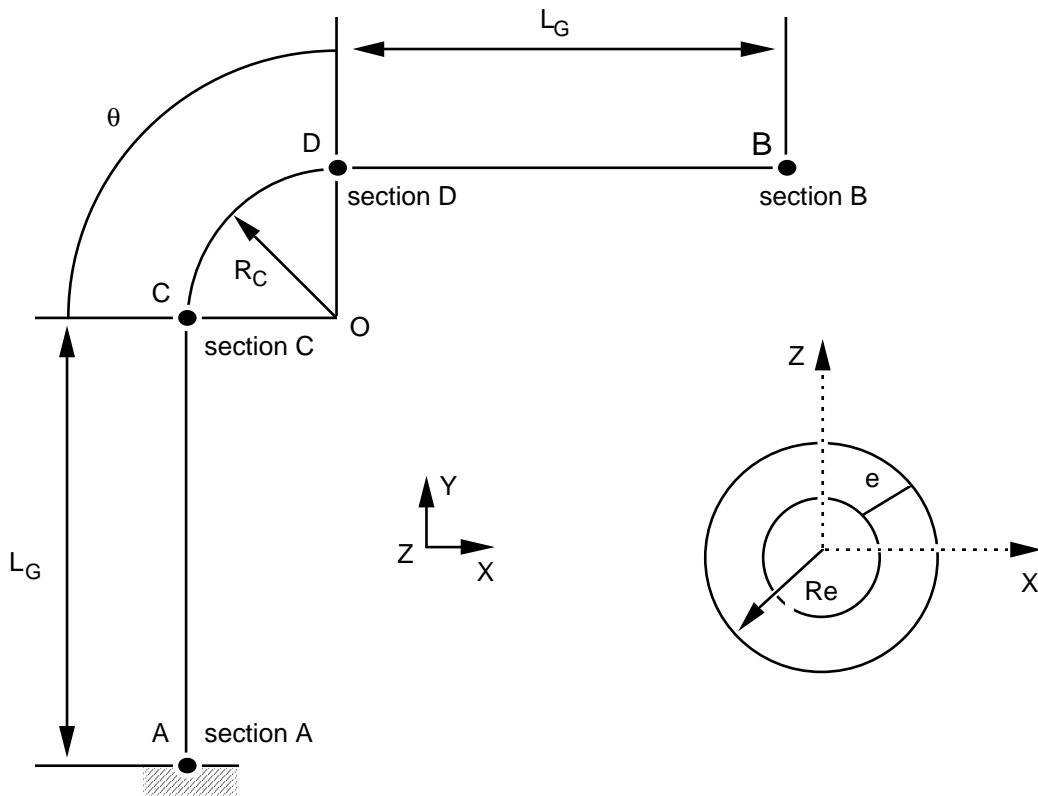


Figure 1.1-a

Note:

| The model is symmetrical along the (A,X,Y) plane.

1.2 Loading

The boundary conditions are the following:

- The pipe is imbedded in the ground at section A,

The load is constant $F_Y = 100\,000\text{ N}$ and is applied to section B along the Y axis.

1.3 Material characteristics

The material properties are that of A42 steel:

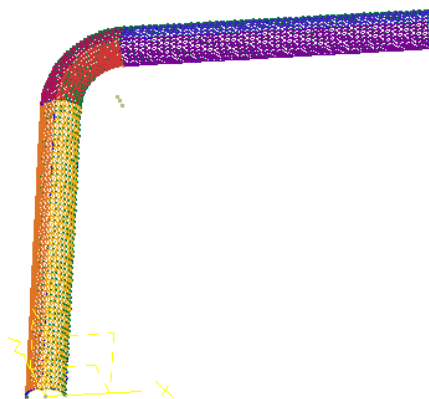
- The Young modulus $E = 204\,000 \cdot E+6\text{ N/m}^2$,
- The Poisson coefficient $\nu = 0.3$.

2 Problem modelling

We can model the problem using DKT shell elements.

2.1 GMSH Meshing

When creating a model using shell elements, the meshing is built by discretizing the average surface of the pipework. Because the geometry is symmetrical along the (A,X,Y) plane, we will only mesh half a surface. The mesh shall need to be sufficiently fine so as to obtain a precise solution (the DTK elements with three nodes having an interpolation of degree 1 in membrane).



Title : An operating example for Code_Aster : calculation of a bent pipe
Author(s) : **M. ABBAS, J.M. PROIX**
Translator(s) : **C. LUZZATO**

Date : 18/05/05
key : U1.05.01-B Page : 4/10

Here we describe the `geo` file from GMSH which will create this mesh:

```
////////////////////////////////////  
// Meshing of bent pipe gmsH 1.60  
////////////////////////////////////  
// Variables  
Rext = 0.2 ;  
Ep = 0.02 ;  
Rm = Rext - (Ep/2.) ;  
RC = 0.6 ;  
LG = 3.0 ;  
  
h = 0.04 ;  
  
Point(1) = {RC, LG, 0., h};  
Point(2) = {RC, LG, 0.1, h};  
Point(3) = {(-1*Rm), 0, 0, h};  
Point(4) = {0, 0, Rm, h};  
Point(5) = {Rm, 0, 0, h};  
Point(6) = {0, 0, 0, h};  
  
Circle(1) = {3,6,4};  
Circle(2) = {4,6,5};  
  
// 1st strait pipe  
Extrude Line {2, {0,LG,0}}  
              {Layers{50,90,1}; };  
Extrude Line {1, {0,LG,0}}  
              {Layers{50,91,1}; };  
  
// Bend  
Extrude Line {3, {0,0,1}, {RC,LG,0.}, -(Pi/2)}  
              {Layers{30,93,1}; };  
Extrude Line {7, {0.,0.,1.}, {RC,LG,0.}, -(Pi/2)}  
              {Layers{30,94,1}; };  
  
// 2nd strait pipe  
Extrude Line {11, {LG,0,0}}  
              {Layers{50,95,1}; Recombine; };  
Extrude Line {15, {LG,0,0}}  
              {Layers{50,96,1}; Recombine; };  
  
Coherence ;  
  
Physical Line(27) = {2,1};  
Physical Line(28) = {23,19};  
Physical Line(29) = {24,16,8,5,13,21};  
Physical Surface(30) = {90,91,93,94,95,96};  
Physical Point(31) = {3};
```

2.2 The Code_Aster commands

The straight pipes and the bend will be modelled using DTK shell elements.

The pipework is propped at the base, for all the nodes in the $Y=0$ plane. The pipework is symmetrical along the Z axis.

- A divided load F^* defined along the Y axis is applied to section B (it is defined so that $2\pi R_{\text{moy}} F^*$ = the total force that we wish to apply).

We shall compute the constraint field for the node elements (`SIGM_ELNO_DEPL`), for each load case. Use `NIVE_COUCHE` to define the calculation level along the thickness

The main calculation steps in Aster are:

- Meshing.
- Defining the finite elements that will be used (`AFFE_MODELE`).
We shall use the mesh element groups created during the meshing.
- Defining and assigning the material (`DEFI_MATERIAU` and `AFFE_MATERIAU`).
The mechanical characteristics are identical for the whole structure.
- Assigning the characteristics for the shell elements (`AFFE_CARA_ELEM`) including the thickness and v vector which define local coordinate system for result analysis (key word `ANGL_REP`). For example, we can choose $V=Oz$.
- Defining boundary conditions and loading (`AFFE_CHAR_MECA`).
- Resolving the elastic problem for each load case (`MECA_STATIQUE`).
Calculating the constraint field at the nodes for each load case ('`SIGM_ELNO_DEPL`' option).
- Printing the results (`IMPR_RESU`).
The average displacement along the B section and the maximum values of the constraint tensor will be printed as a list.

2.3 Analysis of commands

We are now going to list and analyse the commands which are used in such a calculation.

```
Command file

# TITLE PIPEWORK CONTAINING A BEND
# MODEL GENERATED USING DTK SHELL
ELEMENTS
#           PRODUCED BY GMSH

DEBUT ( ) ;
PRE_GMSH( ) ;
MAIL = LIRE_MALLAGE ( ) ;

# Defining the finite elements that
will be used

MODMECA=AFFE_MODELE(MALLAGE=MAIL,

AFFE=_F(GROUP_MA=( 'GM30', 'GM28', ),
PHENOMENE='MECANIQUE',
MODELISATION='DKT', ), );

# Orientation of the GM30 shell
normals

MAIL=MODI_MALLAGE(reuse =MAIL,
                  MALLAGE=MAIL,
                  ORIE_NORM_COQUE=_F(
                    GROUP_MA='GM30',

VECT_NORM=(1.0,0.0,0.0, ),
          GROUP_NO='GM31', ),
          MODELE=MODMECA, );

# Defining the material

ACIER=DEFI_MATERIAU(ELAS=_F(E=20400000
0000.0,

NU=0.3, ), );

CHMAT=AFFE_MATERIAU(MALLAGE=MAIL,

AFFE=_F(TOUT='OUI',

MATER=ACIER, ), );

# Shell characteristics
CARA_COQ=AFFE_CARA_ELEM(

          MODELE=MODMECA,
          COQUE=_F(

GROUP_MA=( 'GM30', 'GM28', ),
```

Analysis

Comments are preceded by the # sign,

Mandatory starting command...
The mesh is in the GMSH format
Reading of the mesh from the mesh
file, and creation of the MAIL concept
which contains the mesh in the Aster
format

A model is a concept containing the
finite element types which will be used
in the calculation

Assigns the mesh elements from the
GM28 and GM30 groups ...

...to mechanical finite elements of the
DSK shell type

Modification of the MAIL mesh

By orienting the normals from group
GM30
So that they follow the (1,0,0) normal

Defined on the GM31 node
within MODMECA model

The characteristics of each material
which constitutes the mesh are given
Young modulus and Poisson coefficient

On the MAIL mesh
And to all mesh elements

We assign the ACIER (steel)
material

We change the elementary
characteristics
On the MODMECA model
Of the shells
Defined in groups GM30 and GM28

Title : An operating example for Code_Aster : calculation of a bent pipe
 Author(s) : **M. ABBAS, J.M. PROIX**
 Translator(s) : **C. LUZZATO**

Date : 18/05/05
 key : U1.05.01-B Page : 7/10

```

                                EPAIS=0.02,

ANGL_REP=(0.0,90.0, , , );

# Defining the boundary conditions

BLOCAGE=AFFE_CHAR_MECA(MODELE=MODMECA,
                        DDL_IMPO=(
                            _F(GROUP_MA='GM27',
                                DX=0.0,
                                DY=0.0,
                                DZ=0.0,
                                DRX=0.0,
                                DRY=0.0,
                                DRZ=0.0, ),
                            _F(GROUP_MA='GM29',
                                DZ=0.0,
                                DRX=0.0,
                                DRY=0.0, ), ), );

# Defining the load
FYTOT = 100000.0;
EPTUB = 0.02;
REXT = 0.2;

RMOY=REXT - EPTUB/2

FYREP=FYTOT/2./PI/RMOY

CHARG1=AFFE_CHAR_MECA(MODELE=MODMECA,
                       FORCE_ARETE=_F(GROUP_MA='GM28',
                                       FY=FYREP, ), );

# Resolution

RESU1=MECA_STATIQUE(

    MODELE=MODMECA
    CHAM_MATER=CHMAT,
    CARA_ELEM=CARA_COQ,

    EXCIT=( _F(CHARGE=BLOCAGE, ),
           _F(CHARGE=CHARG1, ), ), );

# Constraint calculation
RESU1=CALC_ELEM(reuse =RESU1,

OPTION='SIGM_ELNO_DEPL',
          RESULTAT=RESU1, );
  
```

With a shell thickness of 0.2
 In a local coordinate system (useful in post-processing)

For the MODMECA model

The nodes of the mesh element group GM27
 Are imbedded in the ground.

And the nodes of the GM29 mesh element group are defined so that DZ=0, DRX=0 and DRY=0

Defining the total load constant
 Defining the tube thickness constant
 Defining the tube's external radius constant

Calculation of the mean tube radius

Calculation of the total load to be applied

Assigning to model MODMECA a force on segment GM28

Of value FYREP

Global command for the resolution of Static and linear thermo-elastic problems

RESU1 is the name of the result concept

The MODMECA model

the CHMAT material field

The elementary characteristics (shells)

CARA_COQ

The boundary conditions BLOCAGE

The loading CHARG1

reuse=RESU1 means that we « enrich the concept »

within the MODMECA model

With the CHMAT material field

And the elementary characteristics

CARA_COQ

Title : An operating example for Code_Aster : calculation of a bent pipe
Author(s) : M. ABBAS, J.M. PROIX
Translator(s) : C. LUZZATO

Date : 18/05/05
key : U1.05.01-B Page : 8/10

```
# Printing of results so that they can  
be visualised with GMSH
```

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

```
IMPR_RESU(MODELE=MODMECA,  
          FORMAT='GMSH', UNITE=37,  
          RESU=_F(RESULTAT=RESU1,  
                 NOM_CHAM=( 'DEPL' ,  
                             'SIGM_ELNO_DEPL' , , )  
          )
```

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

```
# Creation of a group
```

```
MAIL=DEFI_GROUP(  
              reuse =MAIL,  
  
              MAILLAGE=MAIL,  
              CREA_GROUP_NO=_F(  
                GROUP_MA='GM28' , , )  
              )
```

```
# Creation of a table
```

```
TABDEP1=POST_RELEVE_T(ACTION=_F(  
  
                    INTITULE='DEPB1',  
                    GROUP_NO='GM28',  
                    RESULTAT=RESU1,  
                    NOM_CHAM='DEPL',  
                    TOUT_CMP='OUI',
```

```
OPERATION='MOYENNE' , , )  
# printing of a table
```

```
IMPR_TABLE(TABLE=TABDEP1,
```

```
FILTRE=_F(NOM_PARA='QUANTITE',  
          CRIT_COMP='EQ',  
          VALE_K='MOMENT_0' , , ),  
          NOM_PARA='DY' , , )
```

```
FIN();
```

We calculate 'SIGM_ELNO_DEPL'
meaning
"constraints calculated at the nodes of
each elements by using the
displacements"

Defining of the logical unit for the
GMSH file

We print the results
Derived from the MODMECA model
The results are
In the GMSH format
And are the displacements
Which were printed in the logical unit
associated to the 'POST' file
and result from RESU1
In the GMSH format
and are the constraint at the nodes
Which were printed in the logical unit
associated to the 'POST' file
And result from RESU1

Closing of the logical unit

A new group
reuse=MAIL means that we "enrich"
the mesh concept
From the MAIL mesh
We create a node group
Based on the GM28 mesh elements

We create a table TABDEP1 in post-
processing
Whose name is 'DEPB1'
Which is based on group GM28
And results RESU1
We want the displacements
For all of the components
And the mean

We print the TABDEP1 table
We want the quantity

Which corresponds exactly
To the moment of order 0
on the displacement along y

...
Mandatory command to close an
execution

3 Visualisation with GMSH

With version 7.4 of *Code_Aster*, we can directly print the results in GMSH format. We shall print in the SIGM file bearing the logical unit 37 the constraints (SIYY components only) so that they can be post-processed using GMSH. This component represents the axial component for the whole of the pipe work (because of the orientation chosen in AFFE_CARA_ELEM) :

```
IMPR_RESU(MODELE=MODMECA,  
          FORMAT='GMSH',  
          UNITE=37,  
          RESU=(_F(RESULTAT=RESU1,  
                  NOM_CHAM='SIGM_ELNO_DEPL',  
                  NOM_CMP=('SIXX','SIYY',)),  
              ),  
          )
```

4 Comparison of the obtained results

The results obtained with this model can be compared with those obtained with other models of the same problem:

For the **constant FY load** applied to section B, we compare the displacement at point B for the different models.

The following table gives the values obtained on different models for intermediate mesh refinements.

Constant load FY			
Model	DX	DY	DRZ
Beam flexibility = 1	-2.657E-02	6.702E-02	2.097E-02
Beam RCCM flexibility	-2.983E-02	1.156E-01	3.530E-02
pipe	-2.935E-02	1.083E-01	3.326E-02
Shell (average displacement)	-2.891E-02	1.053E-01	3.242E-02
3D (Average displacement)	-2.907E-02	1.065E-01	-

The following graph presents the deformed shape and the axial constraint iso-values visualised with GMSH.

