CASTEM 2000
THEMATIC CLASSIFICATION
OF ANNOTATED TESTING FILES.

by
E. Le Fichoux
ENSTA - LME

26/01/98

Contents

Chapter 1

ELASTICITY

1.1 Test elas1 Description sheet

Test name elas1

Calculation type MECHANICS ELASTICITY 2D AXI

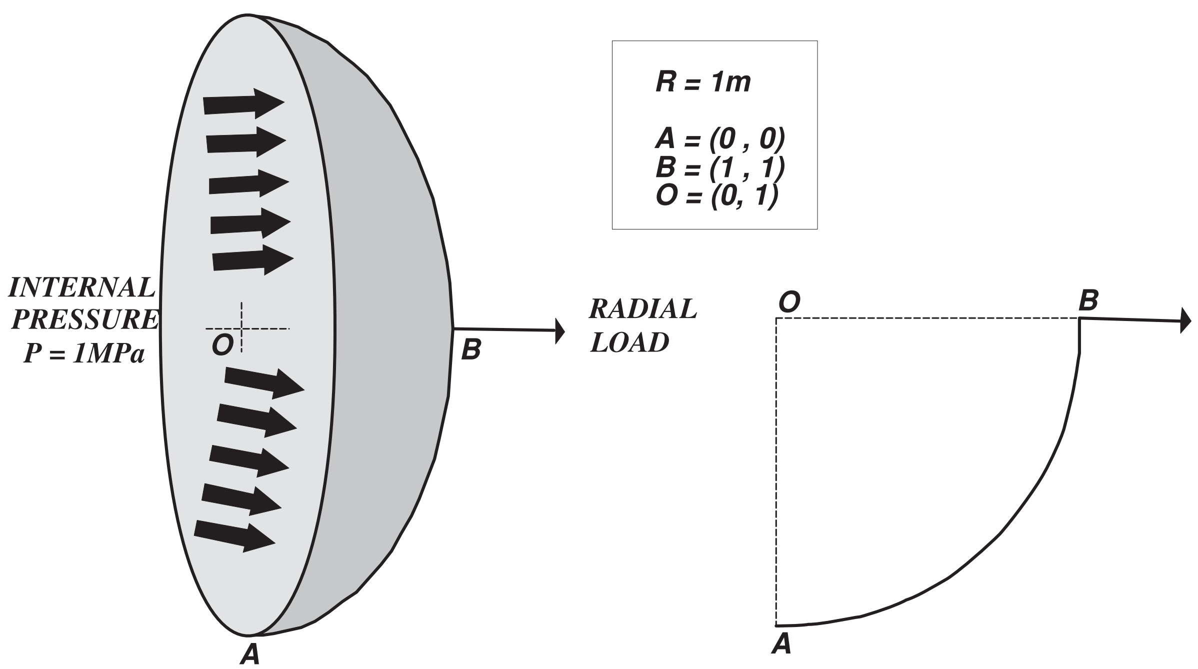
Finite element type COQ2

Topic
A spherical skullcap subjected to different types of load.

The spherical skullcap is subjected to an internal pressure, to its own weight and to a radial load. To solve this problem, we use the principle of superposition.
The equivalent loads for each load were calculated and then added before the resolution of the system. **Goal**

Find the radial displacement at a given point B (1,1).

Reference CASTEM Test NAFEMS : ME1 : Modélisation des structures élastiques dans CASTEM 2000.
Version 97' customer version **Model description**



Solid Representation 3D

Plane Representation 2D

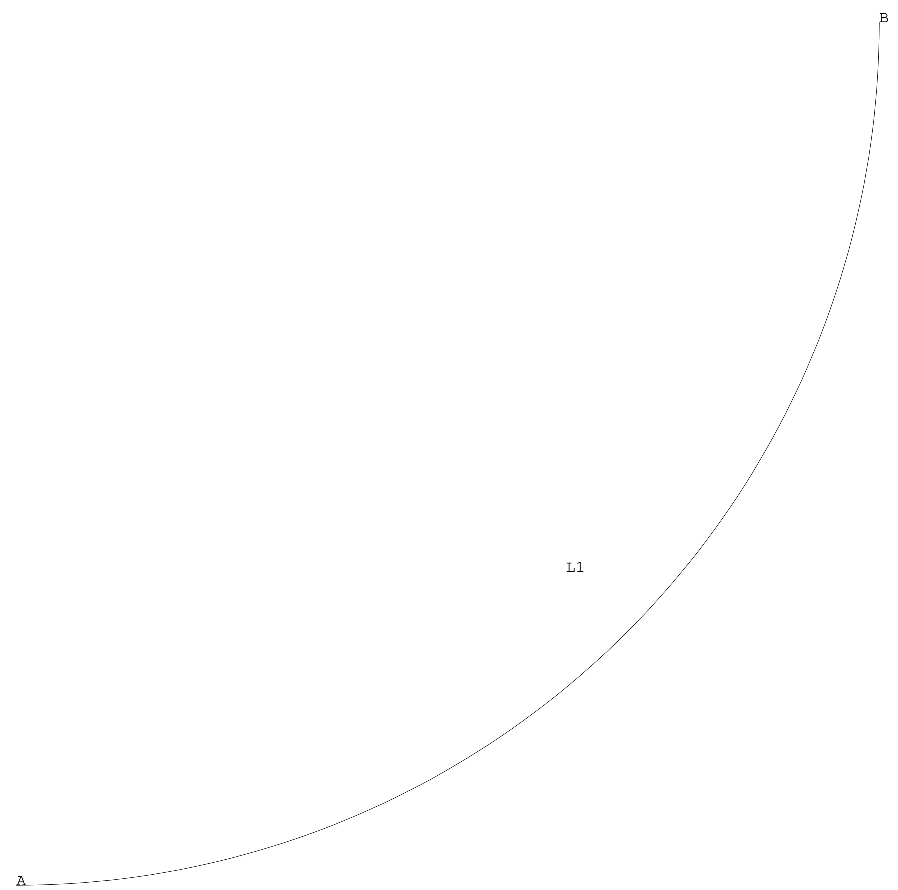
$$
\begin{aligned}
\text{E} &= 2.1\ 10^{11}\ Pa \\
\text{RHO} &= 7.85\ 10^{4}\ g/m^{3} \\
\text{NU} &= 0.28
\end{aligned}
$$

Test elas1 **Results**

RESULTS

	U_r
Reference	$4.677\ 10^{-6}\ m$
Castem	$4.6685\ 10^{-6}\ m$

CASTEM FIGURES



A SPHERIC SKULLCAP SOUBMISSIVE TO DIFFERENT CHARGE TYPE

```
*          Test Elas1.dgibi: Jeux de données        *
*          ---------------------------------        *
*
GRAPH='N';
SAUT PAGE ;
OPTI ECHO 1;
*
*******************************************************
* TEST ELAS1
*
* Une calotte sph}rique est soumise :
*                - @ une pression interne
*                - @ son poids propre
*                - @ une force radiale en tete
*
* En élasticité et en supposant le probleme
* axisymétrique, on se propose de calculer le
* déplacement radial en tete de la calotte.
*
* Pour résoudre ce probleme on utilise le principe
* de superposition. Les forces équivalentes pour chaque
* chargement sont calculées puis ajoutées avant de
* résoudre le systeme.
*
* Le déplacement radial en tete calculé est comparé à
* la valeur théorique égale à 4.677 microns.
*
*******************************************************
*
TITRE 'CALOTTE SPHERIQUE SOUS PLUSIEURS TYPES
DE CHARGEMENT';
OPTION DIME 2 ELEM SEG2 MODE AXIS ;
GRAPH='N';
*
*----------------- DEFINITION DE LA GEOMETRIE --------
*
*
A = 0 0 ;
B = 1 1 ;
O = 0 1 ;
*
L1 = CERC 100 A O B ;
*
SI (NEG GRAPH 'N');
TRACE L1 ;
FINSI;
*
*----- DEFINITION DES CONDITIONS AUX LIMITES ---------
*
CL1 = BLOQ B UZ ;
CL2 = BLOQ A UR ;
CL = CL1 ET CL2 ;
*
* DEFINITION DU MODELE, DU MATERIAU
*   ET DES CARACTERISTIQUES
*
MO = MODL L1 MECANIQUE ELASTIQUE COQ2 ;

MA = MATR MO YOUN 2.1E11 NU 0.3 RHO 7.85E4 ;
CA = CARB MO EPAI 0.02 ;
MA=MA ET CA;
*
*------- CALCUL DE LA MATRICE DE RIGIDITE -------------
*
RI1 = RIGI MO MA ;
RI2 = RI1 ET CL ;
*
*-------------- DEFINITION DES CHARGES ----------------
*
*
*
*------------ PRESSION INTERNE UNIFORME ---------------
*
FO1 = PRES COQU MO -10000 NORM ;
*
* -- POIDS PROPRE (IL FAUT CALCULER LA MATRICE MASSE) -
*
MAS = MASSE MO MA;
EL1 = CHANGE L1 POI1 ;
POP = MANU CHPO EL1 1 UZ -1 ;
FO2 = MAS*POP ;
*
*------------- FORCE RADIALE EN TETE ------------------
*
F=1000*2*PI;
FO3 = FORC FR F B ;
*
FO = FO1 + FO2 + FO3 ;
*
*--CALCUL PAR RESO DES DEPLACEMENTS ET POST-TRAITEMENT-
*
RE = RESO RI2 FO ;
DR = EXTR RE UR B ;
DR = 1000000*DR ;
SAUT PAGE ;
SAUT 2 LIGN ;
MESS ' DEPLACEMENT RADIAL EN TETE   THEORIQUE
CALCULE ' ;
MESS '                              UR=4.677 MICRON
UR =' DR 'MICRON';
TEMPS ;
*--------- CODE DE BON FONCTIONNEMENT -----------------

ERR=100*(ABS(4.677 -DR)/4.677);
SI (ERR < 5);
ERRE 0;
SINON;
ERRE 5;
FINSI;

FIN;
```

Test elas1 **Comments**

1. **Stiffness and Mass Matrix**

   RI1 = RIGI MO MA ;
   RI2 = RI1 ET CL ;
   MAS = MASSE MO MA;

   The operators RIGI and MASS require the mechanical model (here MO) and the material and geometrical characteristics (here MA). For the shell elements it is necessary to define the thickness (EPAI)

2. **Pressure**

   FO1 = PRES COQU MO -10000 NORM ;

   The operator PRES(SION) calculates the nodal forces equivalent to a pressure. The keyword COQU indicates that the pressure is applied on shell elements.
   The operator requires the model (MO) and the value of the pressure. The keyword NORM(AL) indicates that the pressure is positive when it is carried by a vector perpendicular to the element, oriented positively.

3. **Own weight**

   MAS = MASSE MO MA;
   EL1 = CHANGE L1 POI1 ;
   POP = MANU CHPO EL1 1 UZ -1 ;
   FO2 = MAS*POP ;

   There are no operators to calculate the own weight. The solution is to multiply the mass matrix (MAS) by an object of CHPOINT type (POP). The POP object defines the value of gravity (direction UZ and value -1).

4. **Force**

   F=1000*2*PI;
   FO3 = FORC FR F B ;

   The operator FORCE calculates a force field which is given as a localized force. Here the force is defined by its component FR, its value F and the geometry on wich it is applied (point B).

5. **Addition of the three loads**

   FO = FO1 + FO2 + FO3 ;

   The three loads are objects of CHPOINT type. It is also possible to add them in order to get the complete load.

1.2 Test elas2 Description sheet

Test name elas2

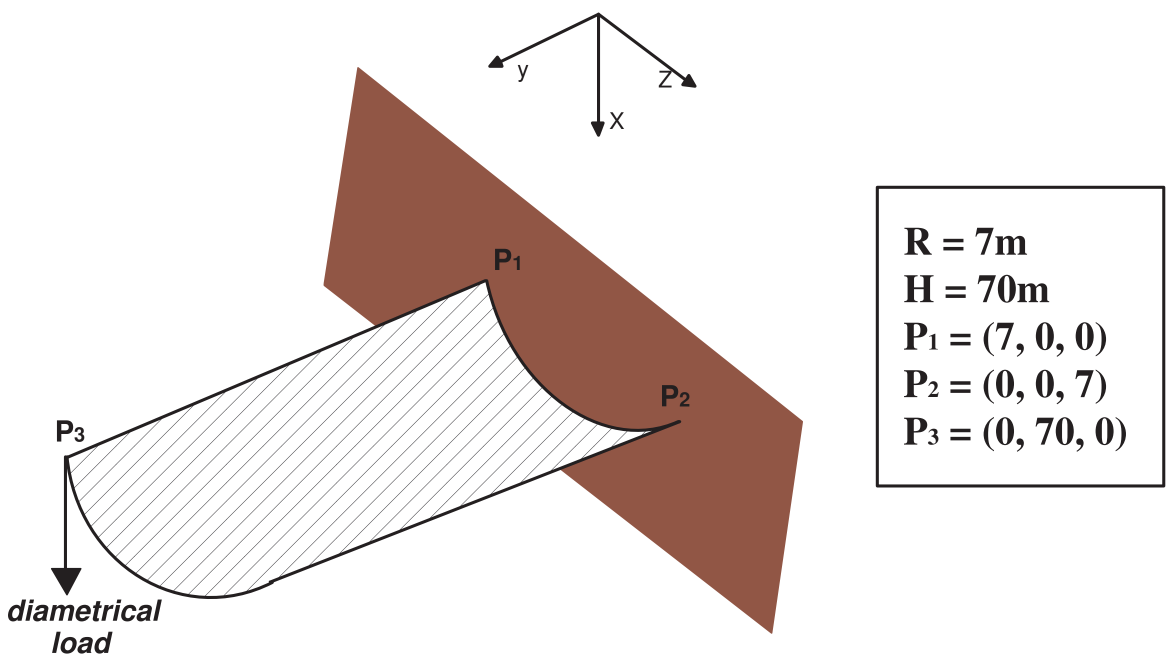
Calculation type MECHANICS ELASTICITY 3D

Finite element type DKT

Topic
A cylindrical solid embedded, subjected to a diametrical load .

The cylindrical structure is pinched at one end while the other one is embedded.
For symmetry reasons, just 1/4 of this structure is modelled. The boundary conditions should reproduce the symmetry.
A diametrical load is applied. **Goal** Find the displacement on X at the point P1 (7,0,0) and the displacement on Z at point P3 (0,70,0) where the diametrical force is applied.

Reference CASTEM Test NAFEMS : ME6 : Modélisation des structures élastiques dans CASTEM 2000.
Version 97' customer version **Model description**



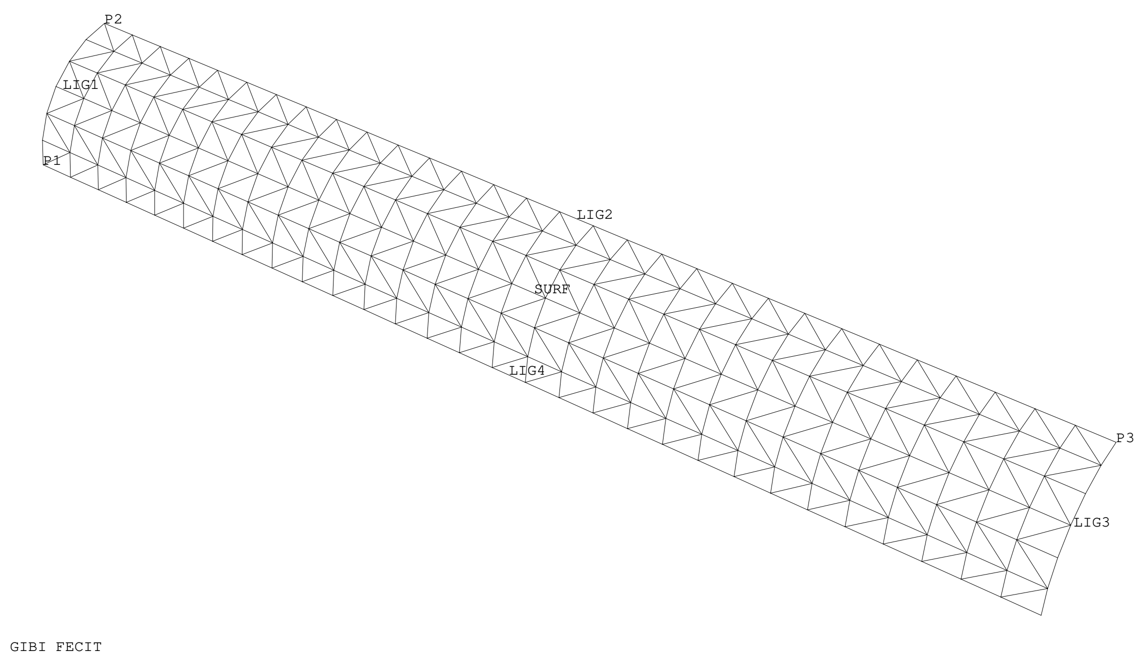
$$\begin{aligned} E &= 3\ 10^7\ Pa \\ RHO &= 7.85\ 10^4\ g/m^3 \\ NU &= 0.3 \end{aligned}$$

Test elas2 **Results**

RESULTS

	U_x (P_1)	U_z (P_3)
Reference	0 cm	-1.14 Cm
Castem	$-2.1247\ 10^{-19}$ Cm	-1.1394 cm

CASTEM FIGURES



```
*          Test Elas2.dgibi: Jeux de données        *
*          ---------------------------------        *
*
******************************************************
OPTION ECHO 0;
GRAPH='N';
SAUT PAGE ;
******************************************************
*   TEST ELAS2
*
* Un cylindre est pincé à une de ses extrémités,
* l'autre extrémité étant encastrée.
*
* Pour des raisons de symétrie seul 1/4 de la structure
* a est modélisé.
*
* Les conditions aux limites sont imposées pour
* respecter les symétries.
* Une force ponctuelle est appliquée diametralement.
*
* Le déplacement en X d'un point de la base est comparé
* au déplacement théorique (déplacement nul puisque
* encastré).
* Le déplacement en Z du point d'application de force
* est comparé à la valeur théorique (-1.140)
*
******************************************************
*
OPTION DIME 3 ELEM TRI3 ;
OPTION ECHO 1 ;
TITRE 'CYLINDRE PINCE A BORDS ENCASTRES' ;
TEMPS ;
*-----------------------------------------------------
* DEFINITION DE LA GEOMETRIE
*-----------------------------------------------------
P1=7 0 0 ;P2=0 0 7 ;C1=0 0 0 ;VECT=0 70 0 ;
LIG1=C 6 P1 C1 P2 ;SURF=LIG1 TRAN 30 VECT ;
LIG1 LIG2 LIG3 LIG4=COTE SURF ;P3=LIG2 POIN FINA ;
*
OEIL1 = 100 100 100;
SI(NEG GRAPH 'N');
    TRAC OEIL1 SURF;
FINSI;
*-----------------------------------------------------
* DEFINITION DU MODELE
*-----------------------------------------------------
MOD1=MODL SURF MECANIQUE ELASTIQUE DKT ;
MAT1=MATR MOD1        YOUN 3.E7   NU 0.3 ;
EPEE=CARB MOD1        EPAI 0.1 ;
MAT1=MAT1 ET EPEE;
*-----------------------------------------------------
*** CALCUL RIGIDITE ET MASSE ;
*-----------------------------------------------------
*
RIG1=RIGI MOD1 MAT1;
*
*-----------------------------------------------------
* CONDITIONS AUX LIMITES
*-----------------------------------------------------
ENC1=BLOQ LIG4 UZ ;ENC2=BLOQ LIG4 RX ;ENC3=BLOQ LIG4 RY ;
ENC4=BLOQ LIG2 UX ;ENC5=BLOQ LIG2 RY ;ENC6=BLOQ LIG2 RZ ;
ENC7=BLOQ LIG3 UY ;ENC8=BLOQ LIG3 RX ;ENC9=BLOQ LIG3 RZ ;
ENC10=BLOQ LIG1 DEPL ROTA ;
ENC11=ENC1 ET ENC2 ET ENC3 ET ENC4 ET ENC5 ET ENC6 ET
ENC7 ET ENC8 ET ENC9 ET ENC10 ;
*------------------------------------------------------
* FORCES
*------------------------------------------------------
F1=0 0 -2500 ;
EFOR=FORC F1 P3 ;
RIG2=RIG1 ET ENC11;
*------------------------------------------------------
* RESOLUTION ET RESULTATS
*------------------------------------------------------
DE1=RESOU RIG2 EFOR;
D1X = EXTR DE1 UX P1 ;
D3Z = EXTR DE1 UZ P3 ;

*------------------------------------------------------
* SORTIE DES RESULTATS
*------------------------------------------------------
SAUT PAGE ;
MESS ' DEPLACEMENT EN X DE LA BASE     VALEUR THEORIQUE :  0.00' ;
MESS ' DEPLACEMENT EN X DE LA BASE     VALEUR CALCULEE  : ' D1X  ;
SAUT 2 LIGNE ;
MESS ' DEPLACEMENT EN Z THEORIQUE :  -1.140' ;
MESS ' DEPLACEMENT EN Z CALCULE   : ' D3Z   ;
TEMPS ;
*              CODE FONCTIONNEMENT
DZREF=-1.14;
RESI=ABS((D3Z-DZREF)/DZREF);
SI(RESI <EG 1E-2);
    ERRE 0;
SINO;
    ERRE 5;
FINSI;
FIN;
```

Test elas2 **Comments**

1. **Resolution**

   DE1 = RESOU RIG2 EFOR;

   The operator RESO(UDRE) constructs the solution of the linear system (Ku=F) with K equal to RIG2, EFOR equal to F and DE1 equal to u.

2. **Results**

   D1X = EXTR DE1 UX P1 ;
   D3Z = EXTR DE1 UZ P3 ;

   The operator EXTR(AIRE) allows to extract a value of one component of an object. Here D1X contains the value of the UX displacement at point P1 and D3Z contains the value of the UZ displacement at point P3.

1.3 Test elas3 Description sheet

Test name elas3

Calculation type MECHANICS ELASTICITY 3D

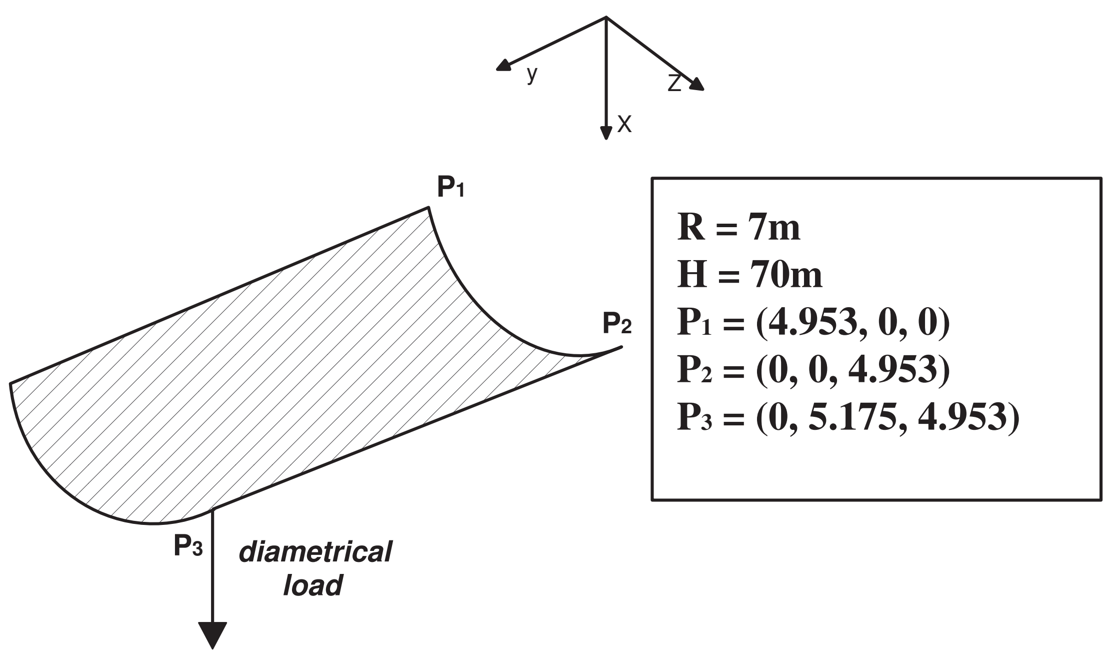
Finite element type TRI3

Topic
The problem is similar to elas2, but the cylinder is free and not embedded. The boundary conditions must reproduce the symmetrical geometry. Just 1/4 of this structure is modelled.
A diametrical load is applied. **Goal** Find the bending in the direction Z at a point P3 (0,70,0), where the

diametrical load is applied. **Reference CASTEM** Test NAFEMS : ME7 : Modélisation des structures

élastiques dans CASTEM 2000. **Version** 97' customer version

Model description



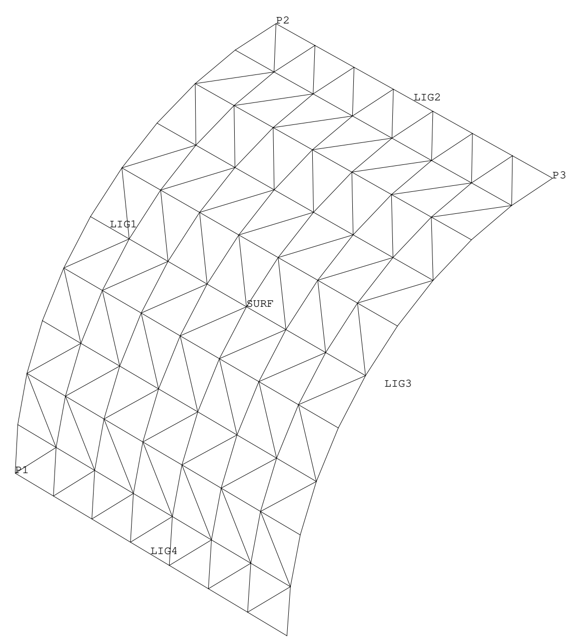
E	=	$3\ 10^7\ Pa$
RHO	=	$7.85\ 10^4\ g/m^3$
NU	=	0.3

Test elas3 **Results**

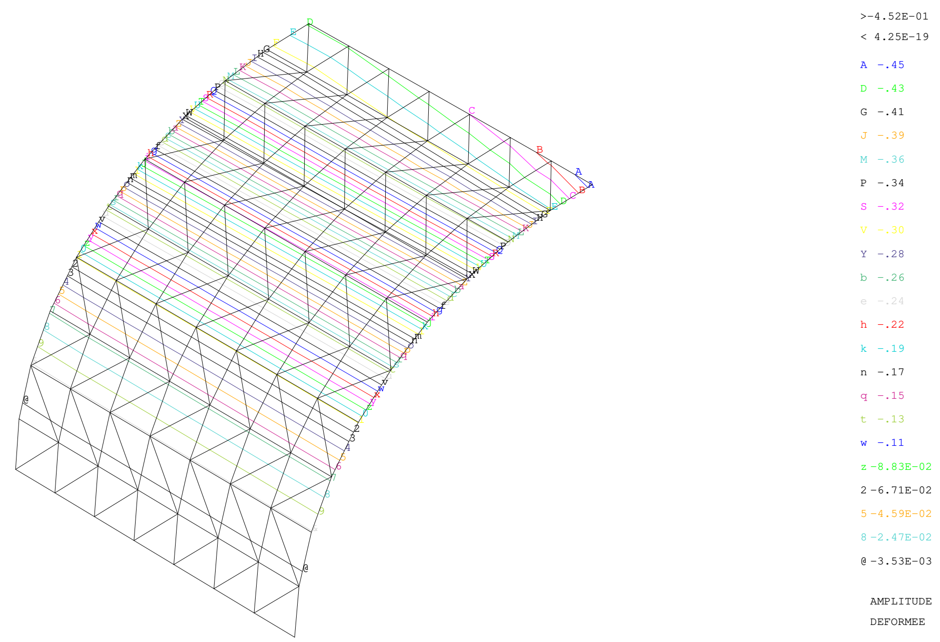
RESULTS

	U_z (P_3)
Reference	-0.452 cm
Castem	-0.45214 cm

CASTEM FIGURES



CYLINDER FREE PINCHED AT BOARDD



CYLINDER FREE PINCHED AT BOARDD

```
*           Test Elas3.dgibi: Jeux de données        *
*           ---------------------------------        *
*
OPTION ECHO 0;
GRAPH='N';
SAUT PAGE ;
*****************************************************
* TEST ELAS3
*
* Meme probleme que DFICH10 mais avec une extrémité
* libre au lieu d'etre encastrée (la géométrie est
* différente).
*
* Un cylindre est pincé à une extrémité, l'autre reste
* libre.
*
* Pour des raisons de symétrie seul 1/4 de la structure
* est discrétisé.
* Les conditions aux limites respectent cette symétrie.
*
* Une force ponctuelle est appliquée diamétralement.
*
* La déflexion suivant Z au point d'application de la
* force est comparée à la déflexion théorique (-0.452).
*
*****************************************************
*
OPTION ECHO 0 ;
OPTION DIME 3 ELEM TRI3 ;DENS 0.75 ;
OPTI ISOV LIGNE;
TITR 'CYLINDRE PINCE A BORDS LIBRES' ;
TEMPS ;
*----------------------------------------------------
* DEFINITION DE LA GEOMETRIE
*----------------------------------------------------
P1=4.953 0 0 ;P2=0 0 4.953 ;C1=0 0 0 ;VECT=0 5.175 0 ;
LIG1=C   P1 C1 P2 ;SURF=LIG1 TRAN    VECT ;
LIG1 LIG2 LIG3 LIG4=COTE SURF ;P3=LIG2 POIN FINA ;
*----------------------------------------------------
* TRACE
*----------------------------------------------------
OEIL1 = 100 100 100;
SI(NEG GRAPH 'N');
   TRAC OEIL1 SURF;
FINSI;
*----------------------------------------------------
* DEFINITION DU MODELE ET DES CARACTERISTIQUES
*----------------------------------------------------

MOD1=MODL SURF MECANIQUE ELASTIQUE DKT;
MAT1=MATR MOD1 YOUN 10.5E6 NU 0.3125 RHO 1. EPAI 0.094 ;

*----------------------------------------------------
* CALCUL RIGIDITE ET MASSE ;
*----------------------------------------------------
*
RIG1=RIGI MOD1 MAT1;
*
*----------------------------------------------------
* CONDITIONS AUX LIMITES
*----------------------------------------------------
ENC1=BLOQ LIG4 UZ ;ENC2=BLOQ LIG4 RX ;ENC3=BLOQ LIG4 RY ;
ENC4=BLOQ LIG2 UX ;ENC5=BLOQ LIG2 RY ;ENC6=BLOQ LIG2 RZ ;
ENC7=BLOQ LIG3 UY ;ENC8=BLOQ LIG3 RX ;ENC9=BLOQ LIG3 RZ ;
ENC10=ENC1 ET ENC2 ET ENC3 ET ENC4 ET ENC5 ET ENC6 ET ENC7 ET ENC8
ET ENC9 ;
*----------------------------------------------------
* FORCE
*----------------------------------------------------
F1=0 0 -100 ;
EFOR=FORC F1 P3 ;
RIG2=RIG1 ET ENC10;
*----------------------------------------------------
* RESOLUTION
*----------------------------------------------------
DE1=RESOU RIG2 EFOR;
D3Z = EXTR DE1 UZ P3;

*----------------------------------------------------
* RESULTATS (isovaleurs de d{placement en Z)
*----------------------------------------------------
SAUT PAGE ;

SURF1=DEFO SURF DE1;
SI(NEG GRAPH 'N');
    TRAC OEIL1 DZ SURF1;
FINSI;
*
MESS ' DEPLACEMENT EN Z THEORIQUE :   -0.452 ';
MESS ' DEPLACEMENT EN Z CALCULE   :   ' D3Z ;
TEMPS ;
*           CODE FONCTIONNEMENT
DZREF=-0.452;
RESI=ABS((D3Z-DZREF)/DZREF);
SI(RESI <EG 5E-2);
     ERRE  0;
SINO;
     ERRE  5;
FINSI;
FIN;
```

Test elas3 **Comments**

1. **Creation of a new field**

   DZ = EXCO UZ DE1;

   The operator EXCO creates, from the field DE1 a new field of the same type by extracting one or more components. Here, the new field DZ is made up of the UZ component of DE1.

2. **Deformed object**

   SURF1 = DEFO SURF DE1;

   The operator DEFO(RME) constructs an object of type DEFORME from an initial geometry (SURF) and a displacement field (DE1).

3. **Isovalues**

   OPTI ISOV LIGNE;
   TRAC OEIL1 DZ SURF1 ;

   The directive OPTI followed by ISOV LIGNE specifies that the operator TRAC will draw the isovalues with lines. The operator TRAC represents the field DZ on the deformed object SURF1 using lines of isovalues.

1.4 Test elas4 Description sheet

Test name elas4

Calculation type MECHANICS ELASTICITY 3D

Finite element type COQ4

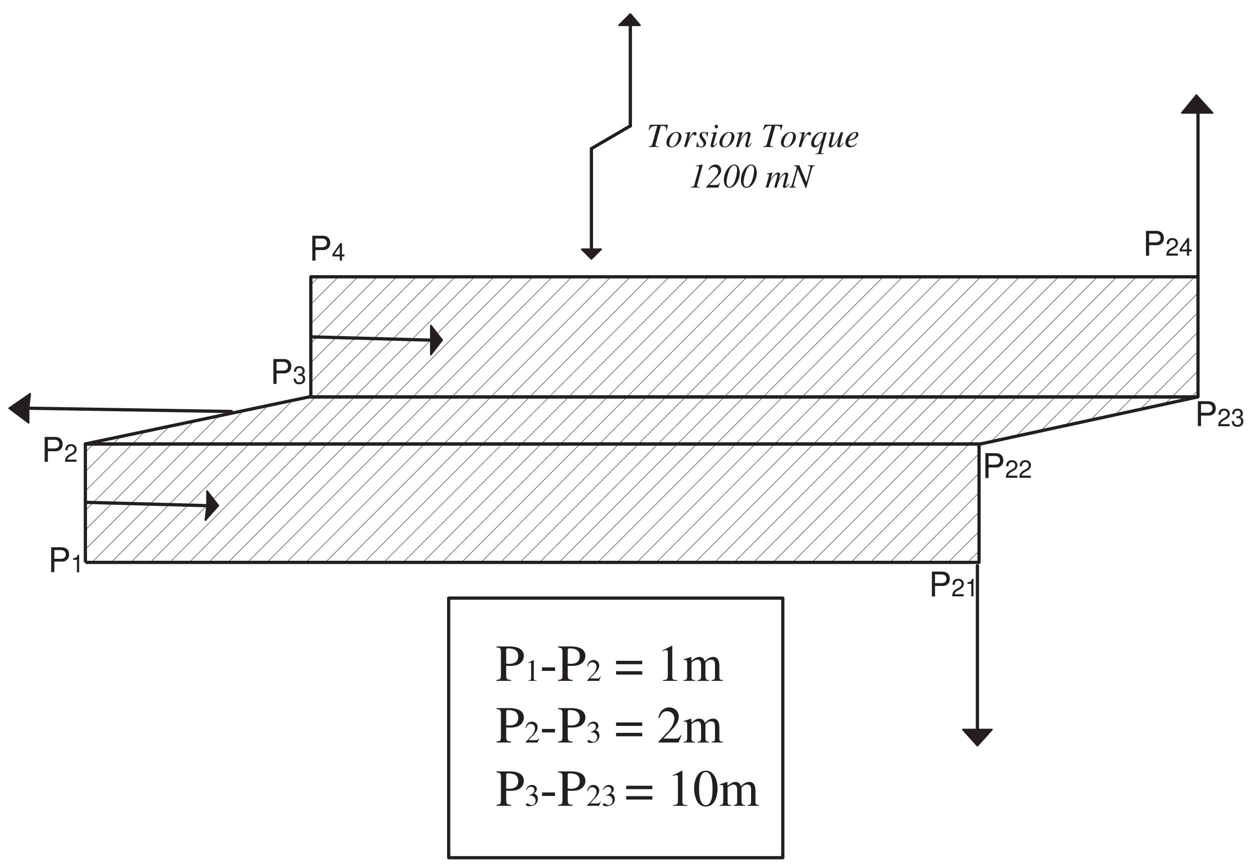
Topic A cantilever beam wich cross section is subject to a torsion torque.

Consider a simple cantilever beam with a cross section. This structure is subjected to a torsion torque. The value of the torsion torque is 1200mN, obtained from two loads of 600N each. The loads are uniformly applied on each side.

Goal Find the axial stress at point A(2.5, 0, 0). The analytical value is Smxx = -108 MPa.

Reference CASTEM Test NAFEMS : ME8 : Modélisation des structures élastique dans CASTEM 2000.
Version 97' customer version

Model description



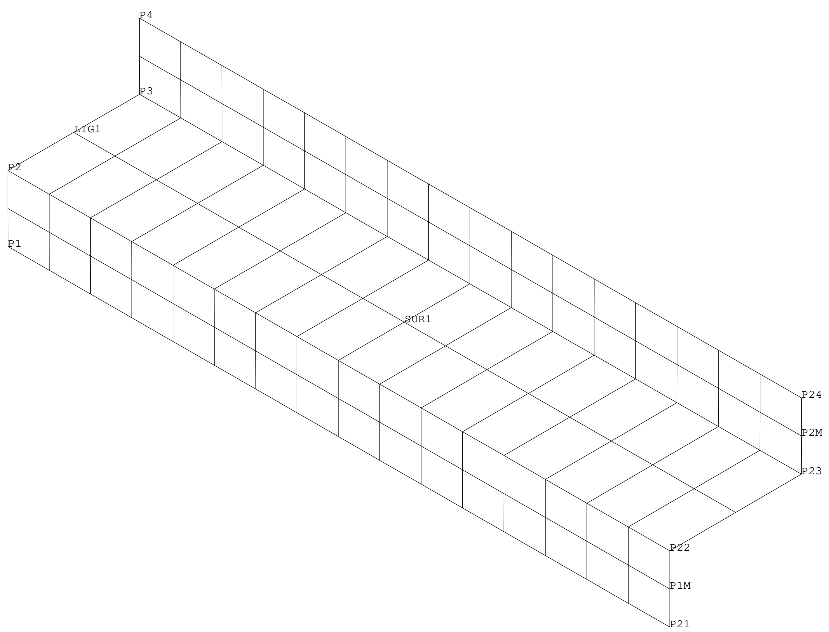
E	=	$210\ 10^9\ Pa$
EPAI	=	$0.1\ m$
NU	=	0.3

Test elas4 **Results**

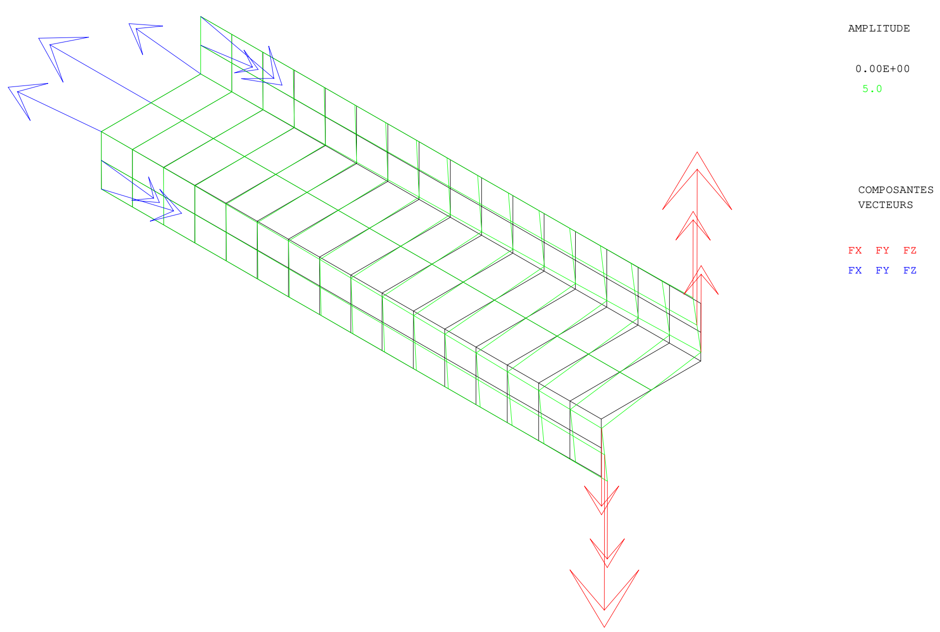
RESULTS

	$Smxx\ (A(2.5,0,0))$
Reference	$-108\ MPa$
Castem	$-112.73\ MPa$

CASTEM figures



TEST ELAS4 : MAILLAGE



ELAS4 : STRAIN, LOADS, REACTIONS

```
*          Test Elas4.dgibi: Jeux de données        *
*          ---------------------------------        *
*
* CAS TEST DU 91/06/13     PROVENANCE : TEST
SAUT PAGE;
*
*******************************************************
*                                                     *
*  TEST ELAS4                                         *
*                                                     *
*  POUTRE A SECTION EN FORME DE "Z" CHARGEE EN TORSION*
*                                                     *
*  Cas test NAFEMS numero LE5 (Z-section cantilever)  *
*                                                     *
*      La poutre est maillee a l'aide d'elements de   *
*  coques COQ4.                                       *
*  Le maillage a ete affine car on ne peut utiliser   *
*  l'operateur CALP avec l'element COQ8.              *
*                                                     *
*      La poutre est soumis a un couple de torsion    *
*  de 1200 m.N, obtenue par deux forces de 600 N      *
*  uniformement reparties sur chaque flanc.           *
*                                                     *
*      On se propose de caculer la contrainte axiale  *
*  de la surface au point A.                          *
*                                                     *
*******************************************************
*
OPTI ECHO 0;
*
OPTI DIME 3;
OPTI ELEM QUA4;
*
GRAPH = 'N';
*
*=====================================================*
*            geometrie : maillage                     *
*=====================================================*
*
* Dimension en metres
*
* Points
*
OEIL = 1000 -1000 1000;
*
P1 = 0 0 0; P21 = (P1 PLUS (10 0 0));
P2 = 0 0 1; P22 = (P2 PLUS (10 0 0));
P3 = 0 2 1; P23 = (P3 PLUS (10 0 0));
P4 = 0 2 2; P24 = (P4 PLUS (10 0 0));
*
P1M = 10 0 .5; P2M = 10 2 1.5;
*
* Droites et lignes
*
* Remarque : ne pouvant utiliser d'element coq8,
* et seulement des coq4 le maillage a ete affine.
*
N1 = 2;
LIG1 = P1 D N1 P2 D N1 P3 D N1 P4;
*
* Maillage
*
N2 = 16;
SUR1 = LIG1 TRANS N2 (10 0 0);
*
ELIM (SUR1 ET P21 ET P22 ET P23 ET P24 ET P1M ET  P2M)
0.001;
*
```

```
SI (NEG GRAPH 'N');
   TITR ' TEST ELAS4 : MAILLAGE ';
   TRAC OEIL QUAL SUR1;
FINSI;
*
*====================================================*
*           modele - materiau - caracteristique      *
*           rigidite - conditions aux limites        *
*====================================================*
*
MODL1 = MODL SUR1 MECANIQUE ELASTIQUE ISOTROPE COQ4;
MATR1 = MATR MODL1 YOUN 210E9 NU 0.3 EPAI 0.1;
RI1 = RIGI MATR1 MODL1;
CL1 = BLOQ DEPL LIG1;
RIG1 = RI1 ET CL1;
*
*====================================================*
*                      Chargement                    *
*====================================================*
*
*   Couple de torsion de 1.2e6 m.N applique dans la
* section x = 10 m  par deux forces uniformement
* reparties egales de 600 N sur chaque flanc.
*
F1  = 0 0 -150E3;
F1M = 0 0 -300E3;
F2  = 0 0  150E3;
F2M = 0 0  300E3;
*
FORC1  = FORC F1  P21;
FORC2  = FORC F1  P22;
FORC1M = FORC F1M P1M;
FORC3  = FORC F2  P23;
FORC4  = FORC F2  P24;
FORC2M = FORC F2M P2M;
*
CHA1 = FORC1 ET FORC2 ET FORC3 ET FORC4 ET FORC1M ET
FORC2M;
*
*====================================================*
*           resolution : champs de deplacements      *
*                        champs de contraintes       *
*====================================================*
*
DEP1 = RESO RIG1 CHA1;
*
* Trace facultatif de la deformee et des reactions
*
SI (NEG GRAPH 'N');
  AMPVEC = .5E-6;
  REA1 = REAC DEP1 RIG1;
  VECT1 = VECT CHA1 (AMPVEC * 20) FX FY FZ ROUGE;
  VECT2 = VECT REA1 AMPVEC FX FY FZ BLEU;
  DEF0 = DEFO 0. DEP1 SUR1;
  DEF1 = DEFO 5. DEP1 SUR1 (VECT1 ET VECT2) VERT;
  TITR ' ELAS4 : Deformee, chargement, reactions';
  TRAC OEIL SUR1 (DEF0 ET DEF1);
FINSI;
*
*====================================================*
*                extraction d'une contrainte         *
*          et comparaison avec solution analytique   *
*====================================================*
*
*     On cherche la contrainte axiale au point A sur la
* surface moyenne.
*
* Calcul des contraintes generalisees dans le repere
```

```
* local des elements
*
CHAM1 = SIGM MODL1 MATR1 DEP1;
*
* Calcul du champs de contraintes au sens des milieux
* continus.
*   Pour cela on utilise l'operateur CALP.
*
CHAM2 = CALP CHAM1 MATR1 MODL1 MOYEN;
CHAM3 = CHAN NOEUD CHAM2 MODL1;
CHPO3 = CHAN CHPO CHAM3 MODL1;
*
PA = SUR1 POIN PROC (2.5 0 0);
SMXXA2 = EXTR CHPO3 SMYY PA;
SMXXA2 = SMXXA2 / 1.E6;
SMXXA1 = -108;
ERGXXA = 100 * (ABS ((SMXXA2 - SMXXA1) / SMXXA1));
*
*=======================================================*
*                  affichage des resultats            *
*=======================================================*
*
MESS ' RESULTATS TEST ELAS4 ';
MESS ' -------------------- ';
SAUT 2 LIGN;
MESS '   On cherche la contrainte axiale sur la surface
moyenne';
MESS ' au point A.';
MESS '   Elle est comparee a une valeur theorique obtenue';
MESS ' analytiquement.';
SAUT 1 LIGN;
SAUT 1 LIGN;
MESS ' Contrainte axiale theorique en A :' SMXXA1 'MPa';
MESS ' Contrainte axiale calculee en A  :' SMXXA2 'MPa';
MESS '    Soit un ecart de :' ERGXXA '%';
SAUT 1 LIGN;
*
ELSUR1 = NBEL SUR1;
NOSUR1 = NBNO SUR1;
*
MESS '    Nombre d elements :' ELSUR1;
MESS '    Nombre de noeuds  :' NOSUR1;
*
*========================================================*
*                 code fonctionnement                  *
*========================================================*
*
*     L'ecart maximum entre valeur theorique et calculee
* doit etre inferieure a 5%.
*
SAUT 2 LIGNE;
SI (ERGXXA <EG 5);
    ERRE 0;
SINON;
    ERRE 5;
FINSI;
*
SAUT 1 LIGN;
TEMPS;
SAUT 1 LIGN;
*
FIN;
```

Test elas4 **Comments**

1. **Strains and reactions**

```
AMPVEC = .5E-6;
REA1 = REAC DEP1 RIG1;
VECT1 = VECT CHA1 (AMPVEC * 20) FX FY FZ ROUGE;
VECT2 = VECT REA1 AMPVEC FX FY FZ BLEU;
DEF0 = DEFO 0. DEP1 SUR1;
DEF1 = DEFO 5. DEP1 SUR1 (VECT1 ET VECT2) VERT;
TITR ' elas4 : STRAIN, LOADS, REACTIONS';
TRAC OEIL SUR1 (DEF0 ET DEF1);
```

The operator REAC(TION) constructs from the solution DEP1 the variation of the second member of the linear system relative to RIG1.
The operator VECT(EUR) constructs an object of VECTEUR type (VECT1 and VECT2) from the components of a field (CHA1 and REA1). The amplication coefficient must be given along with the names of the components (FX, FY, FZ).
The operator DEFO can be used by associating an object of VECTEUR type (VECT1 ET VECT2) with the object of DEFORME type. The result will represent the field of reactions by means of vectors in blue and the field of loads in red on the deformed shape drawing (in green).

2. **Stresses**

```
CHAM1 = SIGM MODL1 MATR1 DEP1;
CHAM2 = CALP CHAM1 MATR1 MODL1 MOYEN;
CHAM3 = CHAN NOEUD CHAM2 MODL1;
CHPO3 = CHAN CHPO CHAM3 MODL1 ;
PA = SUR1 POIN PROC (2.5 0 0);
SMXXA2 = EXTR CHPO3 SMYY PA;
```

The operator SIGMA calculates a stress field (CHAM1) from a displacement field (DEP1). In the case of thin shell elements it is a matter of generalized stresses which are calculated in the local basis.
The operator CALP (CALcul en Peau) calculates a stress field in terms of continuum mechanics starting from a generalized stress field (CHAM1). In the case of shell elements the calculation is carried out at points obtained by projecting supporting points either onto the shell mid-plane (option chosen here with keyword MOYEN) or onto the top or bottom layer.
The operator CHAN(GER) constructs from a field by elements (CHAM2) and from a model object (MODL1) a new field by elements (CHAM3) of the same kind as CHAM2 but born by the nodes (option taken here with keyword NOEUD).
The operator CHAN has different functions, the keyword CHPO loads it to construct from a field by elements and a model object a new field by points (CHPO3).
Note that the names of the components of the first stress field (CHAM1) are different from the names of components of the other stress fields.

3. **Nodes and elements number**

```
ELSUR1 = NBEL SUR1;
NOSUR1 = NBNO SUR1;
```

The operator NBEL supplies the number of elements contained in the geometry SUR1. In the same way, the operator NBNO supplies the number of nodes contained in the geometry SUR1.

1.5 Test elas7 Description sheet

Test name elas7

Calculation type MECHANICS ELASTICITY 2D AXI

Finite element type COQ2

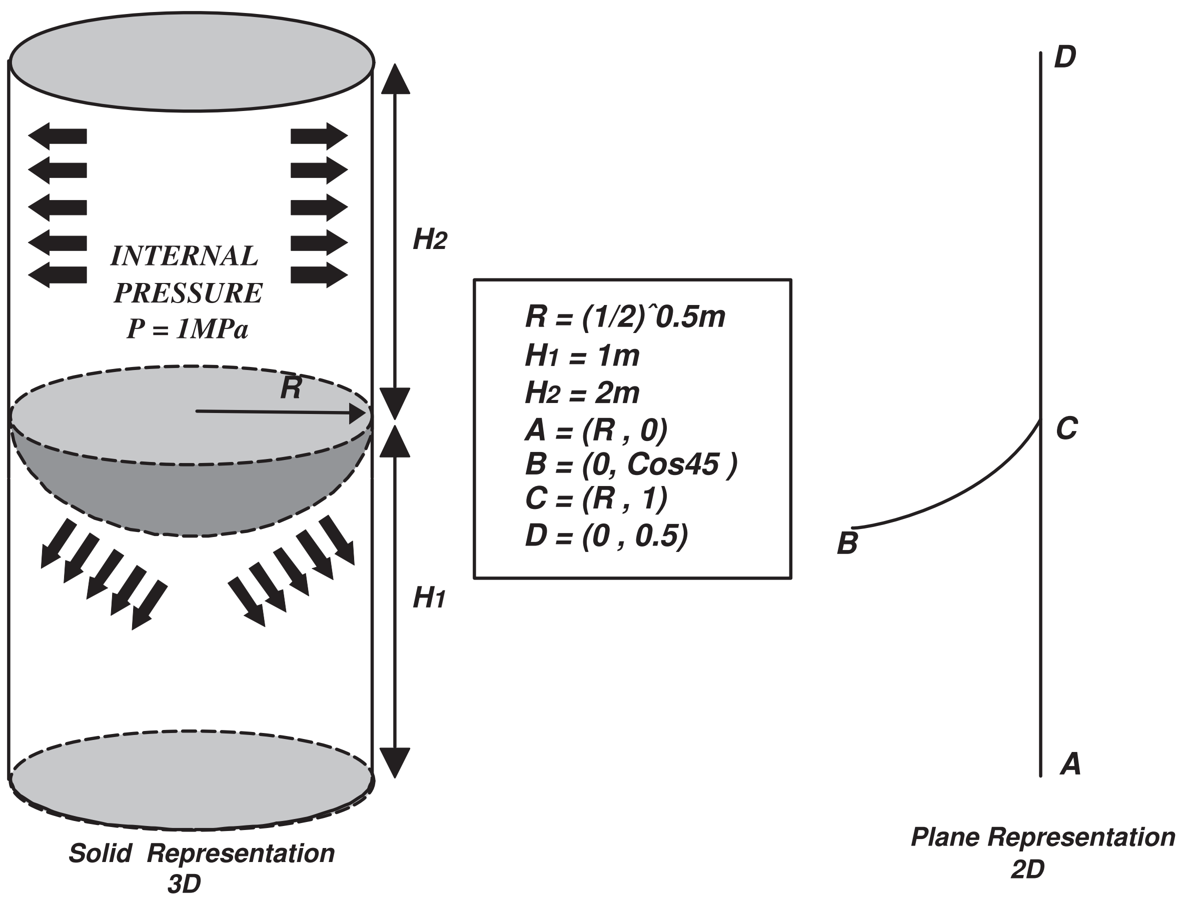
Topic solid : cylinder-sphere-cylinder subjected to an internal pressure.

Consider a solid structure composed by one cylinder surmonted by one sphere and surmonted by one cylinder. This structure is subjected to an internal pressure (P = 1 MPa). On grounds of symmetry (load and geometry), we solve this problem in axisymmetrical model.

Goal Find the axial stress at a point C(0.70, 1). The analytical value is Smzz = -319.9 MPa.

Reference CASTEM Test NAFEMS : ME11 : Modélisation des structures élastiques dans CASTEM 2000. **Version** 97' customer releas

Model description



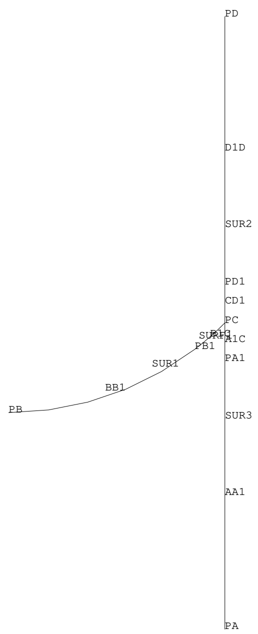
$$
\begin{aligned}
E &= 210\ 10^9\ Pa \\
EPAI &= 0.01\ m \\
NU &= 0.3
\end{aligned}
$$

Test elas7 **Results**

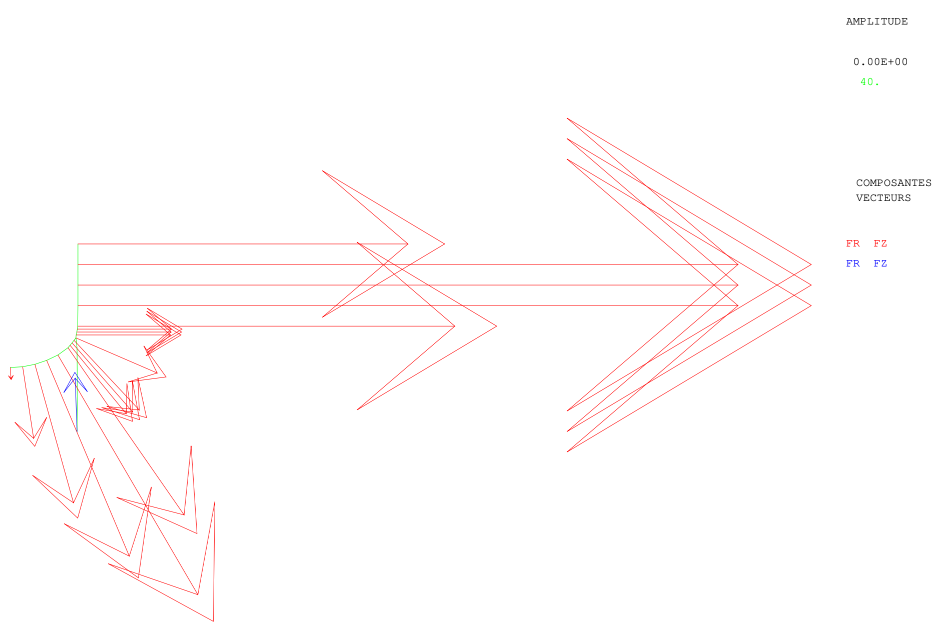
RESULTS

	$Smzz\ (C(0.70, 1))$
Reference	$-319.9\ MPa$
Castem	$-312.29\ MPa$

CASTEM figures



ELAS7 : MESHING



ELAS7 : DEFORMEE, CHARGEMENT, REACTIONS

```
*          Test Elas7.dgibi: Jeux de données        *
*          ---------------------------------        *
*
* CAS TEST DU 91/06/13     PROVENANCE : TEST
SAUT PAGE;
*
*****************************************************
*                                                   *
*   TEST ELAS7                                      *
*                                                   *
*   COQUE AVEC BRANCHEMENT                          *
*                                                   *
*   TEST ELEMENTAIRE NAFEMS : ANALYSE LINEAIRE      *
*                                   ELASTIQUE       *
*          numero LE9               AXISYMETRIQUE   *
*                                                   *
*       L'enceinte est constituee par un cylindre et *
*   par une sphere (demie), reposant sur un cylindre. *
*   L'enceinte est soumise a une pression interne   *
*   uniforme de 1e6Pa.                              *
*      On cherche la contrainte axiale sur la surface *
*   exterieure du cylindre superieure au point C. La *
*   contrainte obtenue est comparee au resultat     *
*   theorique.                                      *
*                                                   *
*****************************************************
*
OPTI ECHO 0;
*
OPTI DIME 2;
OPTI MODE AXIS;
OPTI ELEM SEG2;
*
GRAPH = 'N';
*
*===================================================*
*            geometrie : maillage                   *
*===================================================*
*
* Dimension en metres
*
* Points
*
COS75 = COS 7.5; COS375 = COS 37.5; COS45 = COS 45;
SIN75 = SIN 7.5; SIN375 = SIN 37.5; SIN45 = SIN 45;
R = 2 ** -.5;
*
RA = R; ZA = 0.; PA = RA ZA;
RB = 0; ZB = COS45; PB = RB ZB;
RC = R; ZC = 1.; PC = RC ZC;
RD = R; ZD = 2.; PD = RD ZD;
RO = 0; ZO = 1 + COS45; PO = RO ZO;
PA1 = PC PLUS (0 -0.125);
PD1 = PC PLUS (0 0.125);
PB1 = PB PLUS (SIN375 (1 - COS375));
*
* Droites, cercles et lignes
*
* coefficient de finesse du maillage
*
K = 1;
*
SI (K EGA 1);
  TITR ' LE9 : MAILLAGE GROSSIER';
SINON;
  TITR ' LE9 : MAILLAGE FIN';
FINSI;
*
N1 = 4 * K;
AA1 = PA  DROI N1 PA1;
A1C = PA1 DROI N1 PC;
CD1 = PC  DROI N1 PD1;
D1D = PD1 DROI N1 PD;
B1C = PB1 CERC N1 PO PC;
*
N2 = 5 * K;
BB1 = PB CERC N2 PO PB1;
*
* Maillage
*
SUR1 = BB1 ET B1C;
SUR2 = CD1 ET D1D;
SUR3 = AA1 ET A1C;
SURF1 = SUR1 ET SUR2 ET SUR3;
*
SI (NEG GRAPH 'N');
   TITR 'ELAS7 : MAILLAGE';
   TRAC QUAL SURF1;
FINSI;
*
ELSURF1 = NBEL SURF1;
NOSURF1 = NBNO SURF1;
*
*===================================================*
*   modele - affecte - materiau - caracteristique   *
*       rigidite - conditions aux limites           *
*===================================================*
*
MODL1 = MODL SUR1 MECANIQUE ELASTIQUE ISOTROPE COQ2;
MODL2 = MODL SUR2 MECANIQUE ELASTIQUE ISOTROPE COQ2;
MODL3 = MODL SUR3 MECANIQUE ELASTIQUE ISOTROPE COQ2;
*
MATR1 = MATR MODL1 YOUN 210E9 NU 0.3 EPAI 0.01;
MATR2 = MATR MODL2 YOUN 210E9 NU 0.3 EPAI 0.01;
MATR3 = MATR MODL3 YOUN 210E9 NU 0.3 EPAI 0.01;
*
RI1 = RIGI MODL1 MATR1;
RI2 = RIGI MODL2 MATR2;
RI3 = RIGI MODL3 MATR3;
*
* Conditions aux limites
*    - point pa : encastrement complet.
*
CL1 = BLOQ ROTA DEPL PA;
*
RIG1 = RI1 ET RI2 ET RI3 ET CL1;
*
*===================================================*
*               Chargement                          *
*===================================================*
*
*   Pression interne uniforme de 1e6Pa.
*
CH1 = PRES COQU MODL1 -1.E6 NORM;
CH2 = PRES COQU MODL2 -1.E6 NORM;
CHA1 = CH1 ET CH2;
*
*===================================================*
*       resolution : champs de deplacements         *
*===================================================*
*
DEP1 = RESO RIG1 CHA1;
*
* Trace facultatif de la deformee et des reactions
*
SI (NEG GRAPH 'N');
```

```
  AMPVEC = .4E-6;
  REA1 = REAC DEP1 RIG1;
  VECT1 = VECT CHA1 (AMPVEC * 20) FR FZ ROUGE;
  VECT2 = VECT REA1 AMPVEC FR FZ BLEU;
  DEF0 = DEFO 0. DEP1 SURF1;
  DEF1 = DEFO 40. DEP1 SURF1 (VECT1 ET VECT2) VERT;
  TITR 'ELAS7 : DEFORMEE, CHARGEMENT, REACTIONS';
  TRAC SURF1 (DEF0 ET DEF1);
FINSI;
*
*=====================================================*
*              extraction d'une contrainte            *
*       et comparaison avec solution analytique       *
*=====================================================*
*
*      On cherche la contrainte axiale sur la surface
* exterieure du cylindre superieure au point C.
*
*    Calcul du champs de contraintes generalisees
* uniquement sur le cylindre superieur (sur2).
*
* Reduction du chpoint des deplacements au cylindre
* superieur.
*
DEP2 = REDU DEP1 SUR2;
CHAM1 = SIGM MODL2 MATR2 DEP2;
*
* Calcul du champs de contraintes au sens des milieux
* continus. Pour cela on utilise l'operateur CALP .
*
CHAM2 = CALP CHAM1 MATR2 MODL2 INFE;
*
* Calcul des contraintes aux noeuds du maillage
*
CHAM3 = CHAN NOEUD MODL2 CHAM2;
CHPO3 = CHAN CHPO MODL2 CHAM3;
*
* Extraction de la contrainte recherchee
*
SMZZC = EXTR CHPO3 SMXX PC;
SMZZC = SMZZC / 1.E6;
*
SMZZC1 = -319.9;
*
* Calcul de l'ecart avec la solution de reference
*
ERGZZC = 100 * (ABS ((SMZZC - SMZZC1) / SMZZC1));
*
*=====================================================*
*           affichage des resultats                   *
*=====================================================*
*
MESS ' RESULTATS : TEST ELAS7 ';
MESS ' ---------------------- ';
SAUT 2 LIGN;
*
MESS '   On cherche la contrainte axiale sur la surface
exterieure';
MESS ' du cylindre superieure au point C.';
MESS
' Elle est comparee a une valeur theorique obtenue
analytiquement.';
SAUT 1 LIGN;
MESS ' Contrainte axiale theorique en C:' SMZZC1 'MPa';
SAUT 1 LIGN;
MESS ' Contrainte axiale calculee en C :' SMZZC 'MPa';
SAUT 1 LIGN;
MESS '          Soit un ecart de :' ERGZZC '%';
SAUT 2 LIGN;
*
SI (K EGA 1);
  MESS ' MAILLAGE GROSSIER';
SINON;
  MESS ' MAILLAGE FIN';
FINSI;
*
MESS '    Nombre d elements :' ELSURF1;
MESS '    Nombre de noeuds  :' NOSURF1;
MESS '       soit ' (NOSURF1 * 3) 'd.d.l.';
*
*=====================================================*
*            code fonctionnement                      *
*=====================================================*
*
*  L'ecart maximum entre valeur theorique et calculee
* doit etre inferieure a 5%.
*
SAUT 2 LIGN;
SI (ERGZZC <EG 5);
    ERRE 0;
SINON;
    ERRE 5;
FINSI;
*
*=====================================================*
*          Temps de calcul et fin                     *
*=====================================================*
*
SAUT 1 LIGN;
TEMPS;
SAUT 1 LIGN;
*
FIN;
*
```

1.6 Test elas8 Description sheet

Test name elas8

Calculation type MECHANICS ELASTICITY 2D PLANE STRESS

Finite element type QUA8

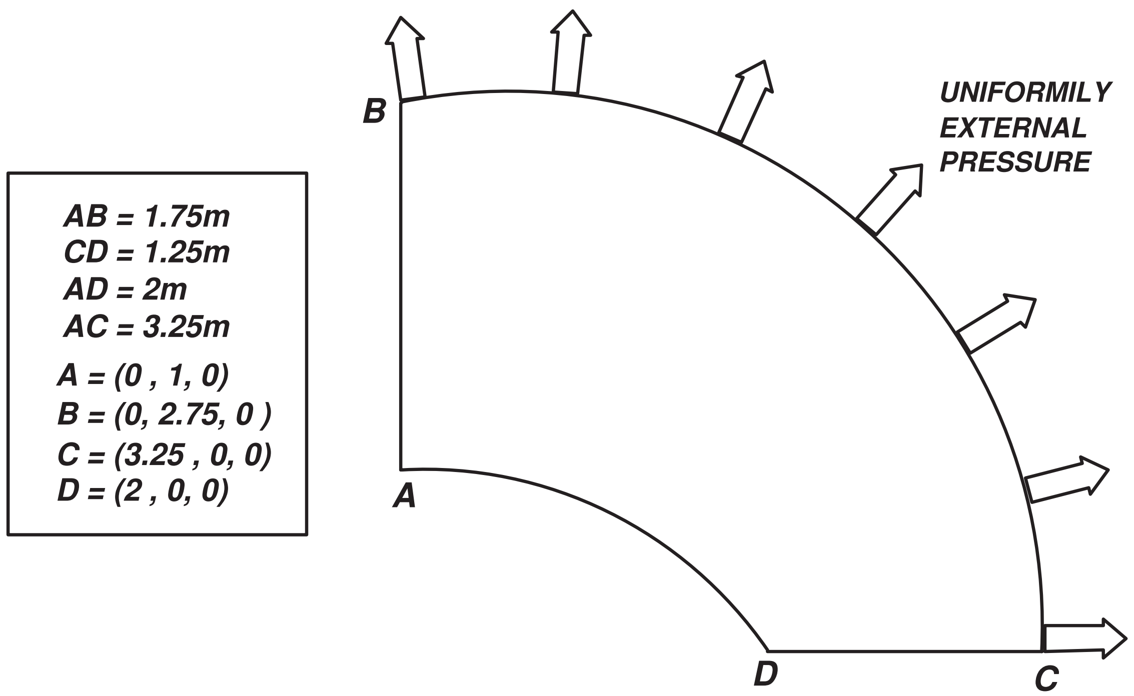
Topic solid : ELLIPTICAL MEMBRANE SUBJECTED TO LINEAR LOAD.

The model is an elliptical membrane, obtained by a projection of one arc of circle onto one plane. This structure is subjected to linear load in one sides. On grounds of symmetry, we solve this problem in axisymmetrical mode.

Goal Find the stress at a point D(2, 0; 0). The analytical value is Smyy = 92.7 MPa.

Reference CASTEM Test NAFEMS : ME12 : Modélisation des structures élastiques dans CASTEM 2000. **Version** 97' customer releas

Model description



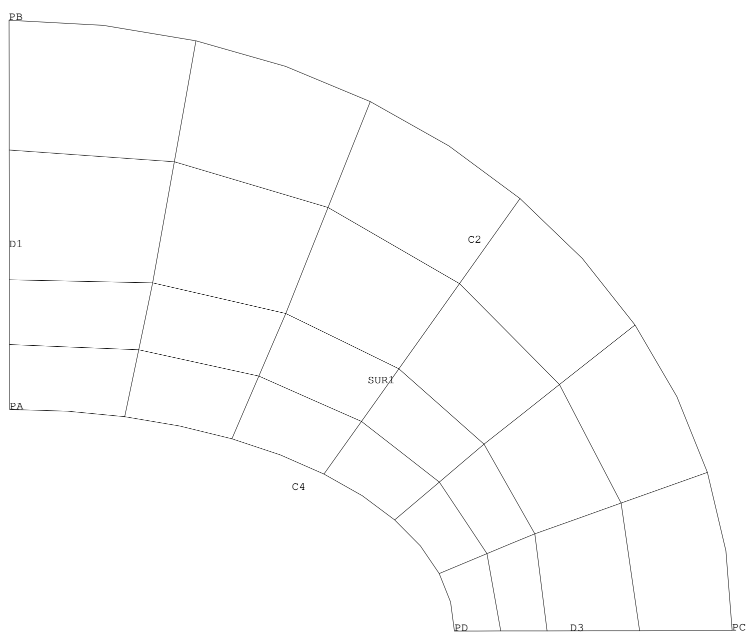
$$E = 210\ 10^9\ Pa$$
$$NU = 0.3$$

Test elas8 **Results**

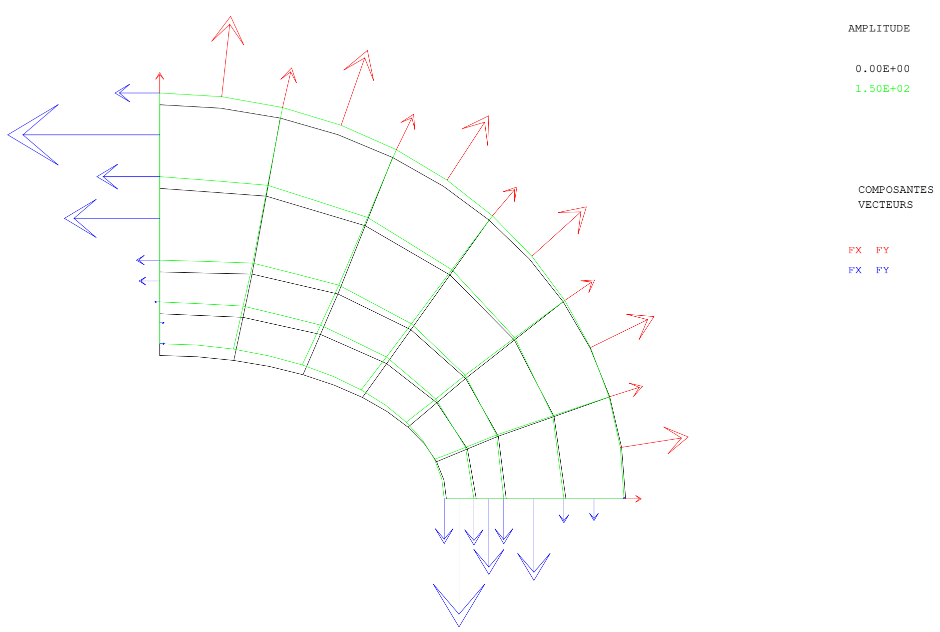
RESULTS

	$Smyy\ (D(2,0,0))$
Reference	$92,7\ MPa$
Castem	$93,548\ MPa$

CASTEM figures



ELAS8 : MESHING



ELAS8 : DEFORMEE, CHARGEMENT, REACTIONS

```
*          Test Elas8.dgibi: Jeux de données        *
*          ---------------------------------        *
*
* CAS TEST DU 91/06/13     PROVENANCE : TEST
SAUT PAGE;
*
*****************************************************
*                                                   *
*  TEST ELAS8                                       *
*                                                   *
*  MEMBRANE ELLIPTIQUE (Contraintes planes)         *
*                                                   *
*  cas-test NAFEMS : test numero LE1                *
*                                                   *
*                                                   *
*    Une membrane elliptique obtenue par projection *
* d'arcs de cercles sur un plan, est soumis a une   *
* force lineique sur l'un de ces cotes.             *
*                                                   *
*    Vue les symetries le probleme est traite en    *
* contraintes planes.                               *
*                                                   *
*    On se propose de calculer la contrainte smyy   *
*    au point D.                                    *
*                                                   *
*****************************************************
*
OPTI ECHO 0;
*
OPTI DIME 3;
OPTI ELEM QUA8;
*
GRAPH = 'N';
*
*===================================================*
*           geometrie : maillage                    *
*===================================================*
*
* Dimension en metres
*
* Points
*
PB = 0. 2.75 0.;
PB1 = 0. 2.75 (((3.25 ** 2) - (2.75 ** 2)) ** 0.5);
PC = 3.25 0. 0.;
*
C1 = PC CERC 6 (0. 0. 0.) PB1;
C2 = C1 PROJ CYLI (0. 0. 1) PLAN (0 0 0) (1 0 0)
(0 1 0);
*
PA = 0. 1. 0.;
PA1 = 0. 1. (((2. ** 2) - (1. ** 2)) ** 0.5);
PD = 2. 0. 0.;
*
C3 = PD CERC 6 (0. 0. 0.) PA1;
C4 = C3 PROJ CYLI (0. 0. 1) PLAN (0 0 0) (1 0 0)
(0 1 0);
*
D1 = PA DROI 2 (0. 1.583 0.) DROI 2 PB;
D3 = PC DROI 2 (2.417 0. 0.) DROI 2 PD;
*
ELIM (D1 ET C2 ET D3 ET C4) 0.0001;
SUR1 = DALL D1 C2 D3 C4 PLAN;
*
SI (NEG GRAPH 'N');
  TITR 'ELAS8 : MAILLAGE';
  TRAC (0 0 1000) QUAL SUR1;
FINSI;
```

```
*
OPTI DIME 2;
OPTI MODE PLAN CONT;
*
*===================================================*
*   modele - affecte - materiau - caracteristique   *
*       rigidite - conditions aux limites           *
*===================================================*
*
MODL1 = MODL SUR1 MECANIQUE ELASTIQUE ISOTROPE QUA8;
MATR1 = MATR MODL1 YOUN 210E9 NU 0.3;
RI1 = RIGI MODL1 MATR1;
*
* Conditions aux limites
*  - droite d1: symetrie par rapport a l'axe des ordo.
*  - droite d2: symetrie par rapport a l'axe des absc.
*
CL1 = SYMT DEPL ROTA (0 0) (0 1) SUR1 0.0001;
CL2 = SYMT DEPL ROTA (0 0) (1 0) SUR1 0.0001;
*
RIG1 = RI1 ET CL1 ET CL2;
*
*===================================================*
*                    Chargement                     *
*===================================================*
*
*   Pression externe uniforme de 10e6Pa.
*
CHA1 = PRES MASS MODL1 -10.E6 C2;
*
*===================================================*
*          resolution : champs de deplacements      *
*===================================================*
*
DEP1 = RESO RIG1 CHA1;
*
* Trace facultatif de la deformee et des reactions
*
SI (NEG GRAPH 'N');
  AMPVEC = 1.E-7;
  REA1 = REAC DEP1 RIG1;
  VECT1 = VECT CHA1 AMPVEC FX FY ROUGE;
  VECT2 = VECT REA1 AMPVEC FX FY BLEU;
  DEF0 = DEFO 0. DEP1 SUR1;
  DEF1 = DEFO 150. DEP1 SUR1 (VECT1 ET VECT2) VERT;
  TITR 'ELAS8 : DEFORMEE, CHARGEMENT, REACTIONS';
  TRAC SUR1 (DEF0 ET DEF1);
FINSI;
*
*===================================================*
*          champs de contraintes                    *
*===================================================*
*
CHAM1 = SIGM MODL1 MATR1 DEP1;
*
* Interpolation aux noeuds du maillage
*
CHAM2 = CHAN NOEUD CHAM1 MODL1;
CHP2 = CHAN CHPO CHAM2 MODL1;
*
*===================================================*
*   extraction et affichage d'une contrainte        *
*===================================================*
*
*   On cherche la contrainte smyy a point D.
*
SMYYD = EXTR CHP2 SMYY PD;SMYYD = SMYYD / 1.E6;
*
```

```
SMYYD1 = 92.7;
ERG = 100 * (ABS ((SMYYD - SMYYD1) / SMYYD1));
*
MESS ' RESULTATS ';
MESS ' --------- ';
SAUT 1 LIGN;
MESS ' CONTRAINTE SMYY THEORIQUE AU POINT D :' SMYYD1
'MPA';
MESS ' CONTRAINTE SMYY CALCULEE AU POINT D  :' SMYYD
'MPA';
MESS '    SOIT UN ECART DE :' ERG '%';
SAUT 1 LIGN;
*
NBNOSUR1 = NBNO SUR1;
NBELSUR1 = NBEL SUR1;
MESS ' NOMBRE D ELEMENTS :' NBELSUR1 'QUA8';
MESS ' NOMBRE DE NOEUDS  ;' NBNOSUR1 ;
SAUT 1 LIGN;
*
*=====================================================*
*          code fonctionnement                        *
*=====================================================*
*
SI (ERG <EG 5.);
   ERRE 0;
SINON;
   ERRE 5;
FINSI;
*
SAUT 1 LIGN;
  TEMPS;
SAUT 1 LIGN;
*
FIN;
```

Test elas8 **Comments**

1. **Projection**

   C4 = C3 PROJ CYLI (0. 0. 1) PLAN (0 0 0) (1 0 0) (0 1 0);

   The operator PROJ(ETER) constructs the object C4 which results from the projection of the object C3 with respect of the vector (0. 0. 1.). The projection is done onto a plane support defined by 3 points (keyword PLAN followed by 3 points).

2. **Boundary conditions of symmetry**

   CL1 = SYMT DEPL ROTA (0 0) (0 1) SUR1 0.0001;
   CL2 = SYMT DEPL ROTA (0 0) (1 0) SUR1 0.0001;

   The operator SYMT imposes boundary conditions of symmetry type on the degrees of freedom in rotation (keyword ROTA). The two points define the axis of symmetry, SUR1 is the geometry on which are imposed the conditions and the real 0.0001 is the criterion of selection of the points.

3. **Pressure**

   CHA1 = PRES MASS MODL1 -10.E6 C2;

   The operator PRES calculates the nodal forces equivalent to the pressure applied on the model object MODL1. The keyword MASS indicates that the type of the elements are massive (QUA4).

1.7 Test elas11 Description sheet

Test elas11 **Description sheet**

Test name elas11

Calculation type MECHANICS ELASTICITY 2D AXI

Finite element type QUA8

Topic A solid (cylinder-cone-sphere), subjected to a gradient linear of temperature

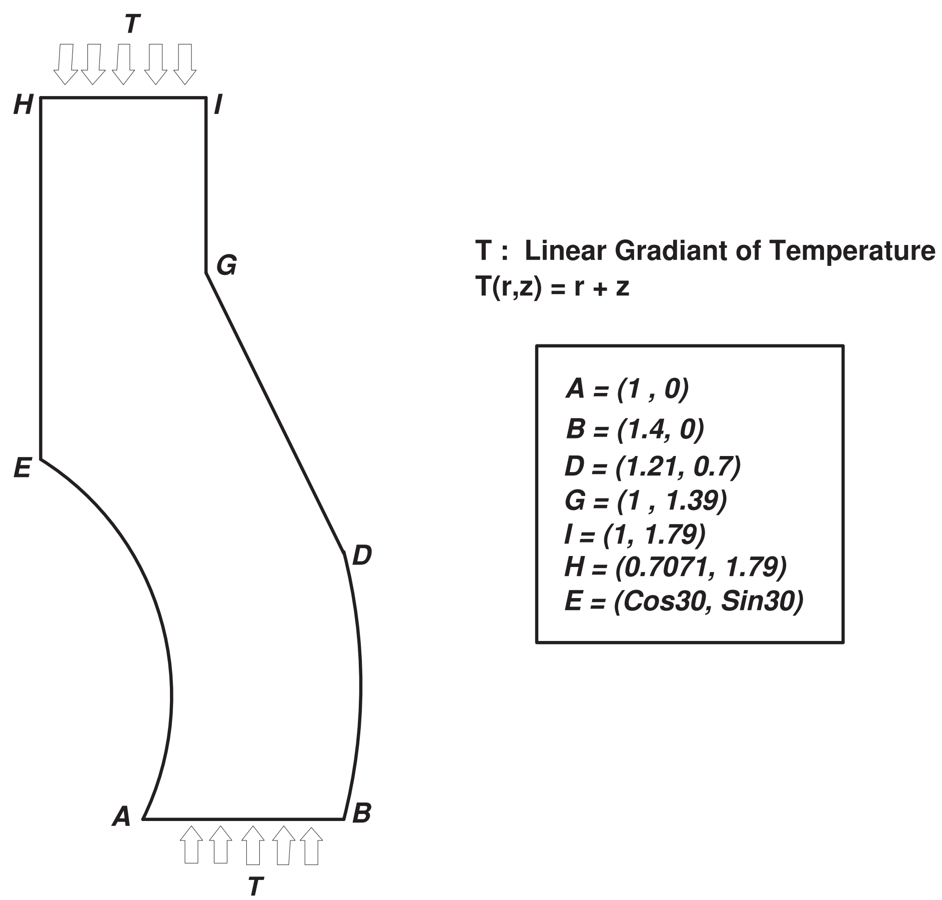
The structure is similar to ELAST10, but we solve this problem in plane axisymmetrical mode. The field of temperature is described by :

$$T(R+z) = R+z$$

Goal Find the stress at a point A(1, 0). The analytical value is Smzz(A) = 105 MPa.

Reference CASTEM Test NAFEMS : LE11 : Modélisation des structures élastiques dans CASTEM 2000. **Version** 97' customer releas

Model description



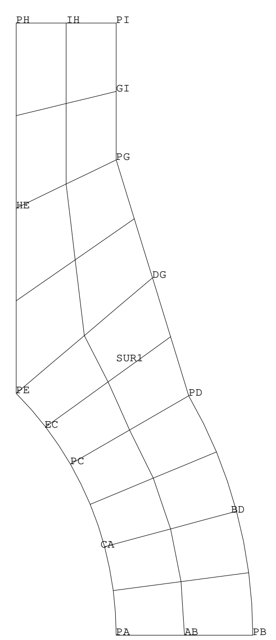
$$
\begin{aligned}
\mathrm{E} &= 210\ 10^{9}\ Pa \\
\mathrm{NU} &= 0.3
\end{aligned}
$$

Test elas11 **Results**

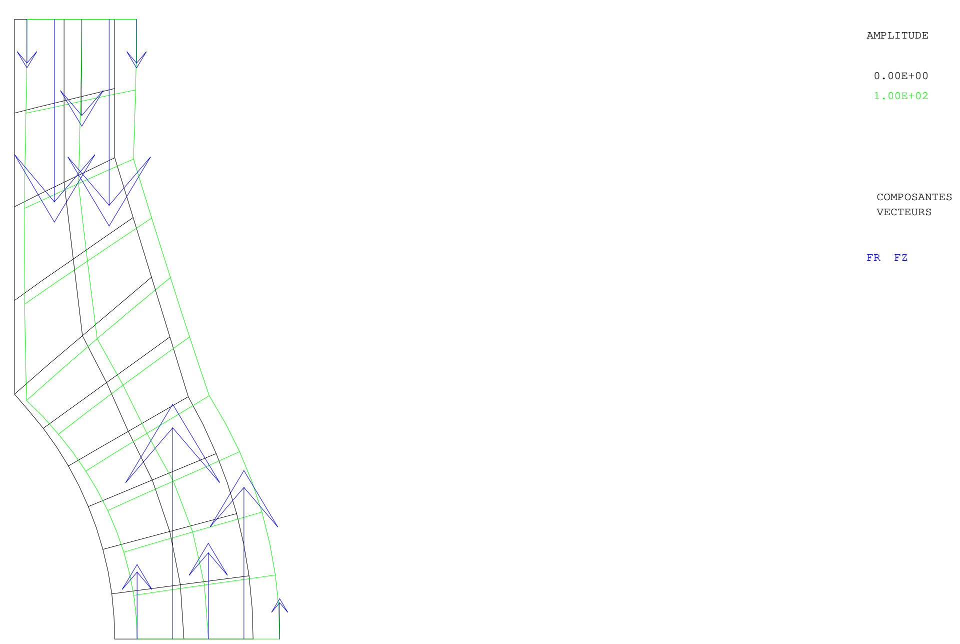
RESULTS

	$Smyy\ (A(1,0))$
Reference	$-105\ MPa$
Castem	$-103.34\ MPa$

CASTEM figures



ELAS11 : FINE MESHING



ELAS11 ; DEFORMEE, REACTIONS

```
*          Test Elas11.dgibi: Jeux de données        *
*          ---------------------------------         *
*
* CAS TEST DU 91/06/13     PROVENANCE : TEST
SAUT PAGE;
*
*******************************************************
*                                                    *
*   TEST ELAS11                                      *
*                                                    *
*        SOLIDE (CYLINDRE - CONE - SPHERE)           *
*           SOUMIS A UN GRADIENT                     *
*    DE TEMPERATURE (Calcul axisymetrique 2D).       *
*                                                    *
*   Test NAFEMS : numero LE11                        *
*   Nom : solid cylinder / taper / sphere temperature *
*                                                    *
*    La structure est une enceinte epaisse.  Elle est *
*   constituee d'une partie spherique surmontee d'une *
*   partie conique, elle -meme surmontee d'une partie *
*   cylindrique.                                     *
*                                                    *
*      Elle est soumis a un gradient lineaire de     *
*   temperature s'exercant radialement et axialement. *
*   Le champs de temperature s'exprime comme suit :  *
*                                                    *
*          T(x,y,z) = (x**2 + y**2)**.5 + z          *
*       ou                                           *
*          T(r,z)   = r + z                          *
*                                                    *
*    L'ensemble est maille avec des elements massifs  *
*   CU20. Le maillage est obtenue par rotation de 90  *
*   degres d'une coupe verticale.                    *
*                                                    *
*    On se propose de calculer la contrainte smyy au  *
*   point A. Cette valeur est ensuite comparee avec  *
*   la valeur de reference obtenue a l'aide d'un     *
*   calcul axisymetrique sur un maillage fin.        *
*                                                    *
*******************************************************
*
OPTI ECHO 0;
*
OPTI DIME 2;
OPTI MODE AXIS;
OPTI ELEM QUA8;
*
GRAPH = 'N';
*
*====================================================*
*            geometrie : maillage                    *
*====================================================*
*
* Dimension en metres
*
* Points
*
PO = 0. 0.;
P1 = 0. 1.;
PA = 1. 0.;
PB = 1.4 0.;
PE = (1. * (COS 30)) (1. * (SIN 30));
PD = (1.4 * (COS 30)) 0.7;
PC = (COS 30) (SIN 30);
PE = (COS 45) (SIN 45);
PH = 0.7071 1.79;
PI = 1. 1.79;
PG = 1. 1.39;
*
* Droites, cercles et lignes
*
* coefficient de finesse du maillage
*
K = 2;
*
N1 = 1 * K;
AB = PA DROI N1 PB;
GI = PG DROI N1 PI;
EC = PE CERC N1 PO PC;
IH = PI DROI N1 PH;
*
N2 = 2 * K;
BD = PB CERC N2 PO PD;
DG = PD DROI N2 PG;
CA = PC CERC N2 PO PA;
HE = PH DROI N2 PE;
*
* maillage par elements massifs du volume
*
SUR1 = DALL AB (BD ET DG ET GI) IH (HE ET EC ET CA)
PLAN;
*
SI (NEG GRAPH 'N');
  SI (K EGA 1);
    TITR 'ELAS11 : MAILLAGE GROSSIER';
  SINON;
    TITR 'ELAS11 : MAILLAGE FIN';
  FINSI;
  TRAC QUAL SUR1;
FINSI;
*
ELSUR1 = NBEL SUR1;
NOSUR1 = NBNO SUR1;
*
*====================================================*
*                  modele - materiau                 *
*              caracteristique - rigidite            *
*====================================================*
*
MODL1 = MODL SUR1 MECANIQUE ELASTIQUE ISOTROPE QUA8;
MATR1 = MATR MODL1 YOUN 210E9 NU 0.3 ALPH 2.3E-4;
RI1 = RIGI MODL1 MATR1;
*
*====================================================*
*              Conditions aux limites                *
*====================================================*
*
*    - deplacement vertical nul pour les faces superieure
*      et inferieure.
*
CL1 = BLOQ UZ AB;
CL2 = BLOQ UZ IH;
*
CLI1 = CL1 ET CL2;
RIG1 = RI1 ET CLI1;
*
*====================================================*
*                 Chargement thermique               *
*====================================================*
*
*   Le solide est soumis a un gradient de temperature
* lineaire, s'appliquant radialement et axialement.
*   La temperature est la suivante :
*
*        T(r,z) = r + z
*
```

```
* Creation du champs de temperature
*
CHPR = COOR 1 SUR1;
CHPZ = COOR 2 SUR1;
*
CHP4 = CHPR + CHPZ;
*
* Transformation du sous-type "scal" en sous-type "t"
*  pour chp4. Utilisation de l'operateur "vari".
*
LIS10 = PROG 0. 200.;
LIS20 = PROG 0. 200.;
EVOL1 = EVOL MANU SCAL LIS10 T LIS20;
CHP5 = VARI CHP4 EVOL1 T;
*
* Creation du champ de contraintes equivalentes
*
CHAM10 = THET MODL1 MATR1 CHP5;
*
* Creation du champ de forces equivalentes
*
CHAR1 = BSIG MODL1 CHAM10;
*
*====================================================*
*       Resolution : champs de deplacements          *
*====================================================*
*
DEP1 = RESO RIG1 CHAR1;
*
* Trace facultatif de la deformee et des reactions
*
SI (NEG GRAPH 'N');
  AMPVEC = 1.E-8;
  REA1 = REAC DEP1 RIG1;
  VECT1 = VECT REA1 AMPVEC FR FZ BLEU;
  DEF0 = DEFO 0. DEP1 SUR1;
  DEF1 = DEFO 100. DEP1 SUR1 VECT1 VERT;
  TITR 'ELAS11 ; DEFORMEE, REACTIONS';
  TRAC SUR1 (DEF0 ET DEF1);
FINSI;
*
*====================================================*
*             extraction d'une contrainte            *
*       et comparaison avec solution analytique      *
*====================================================*
*
*    On cherche la contrainte axiale sigzz au noeud A
*
* Calcul du champs de contraintes totales.
*
CHAM1 = SIGM MODL1 MATR1 DEP1;
*
* On retranche les contraintes d'origine thermique
*
CHAM1 = CHAM1 - CHAM10;
*
* Interpolation aux noeuds du maillage.
*
CHAM2 = CHAN NOEUD MODL1 CHAM1;
CHPO10 = CHAN CHPO MODL1 CHAM2;
*
* Extraction de la contrainte recherchee.
*
SMZZA = EXTR CHPO10 SMZZ PA;
SMZZA = SMZZA / 1.E6;
*
SMZZA1 = -105;
*
* Calcul de l'ecart avec la solution de reference.
*
ERGZZA = 100 * (ABS ((SMZZA - SMZZA1) / SMZZA1));
*
*====================================================*
*           affichage des resultats                  *
*====================================================*
*
MESS ' RESULTATS ';
MESS ' --------- ';
SAUT 1 LIGN;
*
MESS '    La valeur cherchee est la contrainte axiale
au point A,';
MESS
  ' elle est comparee a une valeur theorique obtenue
analytiquement.';
SAUT 1 LIGN;
MESS ' Contrainte verticale theorique en A :' SMZZA1
 'MPa';
SAUT 1 LIGN;
MESS ' Contrainte verticale calculee en A  :' SMZZA
 'MPa';
SAUT 1 LIGN;
MESS '          Soit un ecart de :' ERGZZA '%';
SAUT 2 LIGN;
*
SI (K EGA 1);
  MESS ' Maillage grossier';
SINON;
  MESS ' Maillage fin';
FINSI;
*
MESS '    Nombre d elements :' ELSUR1;
MESS '    Nombre de noeuds  :' NOSUR1;
MESS '                 Soit :' (NOSUR1 * 3) 'd.d.l.';
*
*====================================================*
*                code fonctionnement                 *
*====================================================*
*
*     L'ecart maximum entre valeur theorique et
* calculee doit etre inferieure a 5%.
*
SAUT 1 LIGNE;
SI (ERGZZA <EG 5);
    ERRE 0;
SINON;
    ERRE 5;
FINSI;
*
*====================================================*
*                Temps de calcul et fin              *
*====================================================*
*
SAUT 1 LIGN;
TEMPS;
SAUT 1 LIGN;
*
FIN;
*
```

Test elas11 **Comments**

1. **Creation of a variable field**

   CHP4 = CHPR + CHPZ;
   ...
   EVOL1 = EVOL MANU SCAL LIS10 T LIS20;
   CHP5 = VARI CHP4 EVOL1 T;

   The operator VARI calculates a variable field from the field CHP4 and the variation law in the form of the function described in EVOL1. T will be the name of the component of the generated field CHP5.

2. **Thermal stresses**

   CHAM10 = THET MODE1 MATE1 CHP5;
   CHAR1 = BSIG MODE1 CHAM10;

   The operator THET calcultes the stresses associated with the temperature field CHP5.
   The operator BSIGMA calculates the nodal force field resulting from the integration of the stress field CHAM10. The object CHAR1 represent nodal forces equivalent to the thermic load.

1.8 Test elas13 Description sheet

Test elas13 **Description sheet**

Test name elas13

Calculation type MECHANICS ELASTICITY 3D AXI

Finite element type POUT

Topic BEAM ON ELASTIC GROUND WITH ARTICULATED EXTRIMITIES

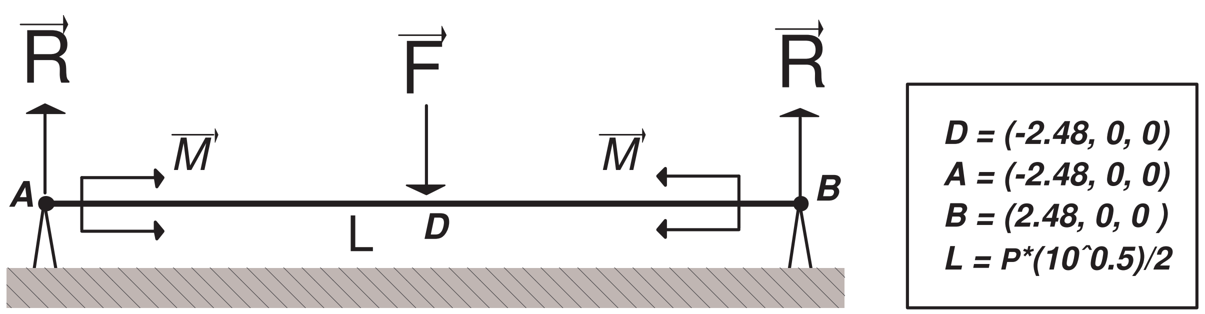
The structure is composed by a simple cantilever beam transported by an elastic ground. This beam is articulated in two extremities .

Goal Find the Rotation and the Reaction at one extremity, point A(-2.48, 0, 0), and the deflexion at D (0, 0, 0) (center of the beam).

Reference CASTEM Test VPCS : SSLL16/89 : Modélisation des structures élastiques dans CASTEM 2000. M. Courtand P. Lebelle Formulaire du béton armé T.2 Paris Eyrolles 1976. **Version** 97'

customer releas

Model description



$$\begin{array}{lcl} \text{E} & = & 2\ 10^{11}\ Pa \\ \text{NU} & = & 0.3 \\ \text{SECT} & = & 1000\ m^2 \end{array}$$

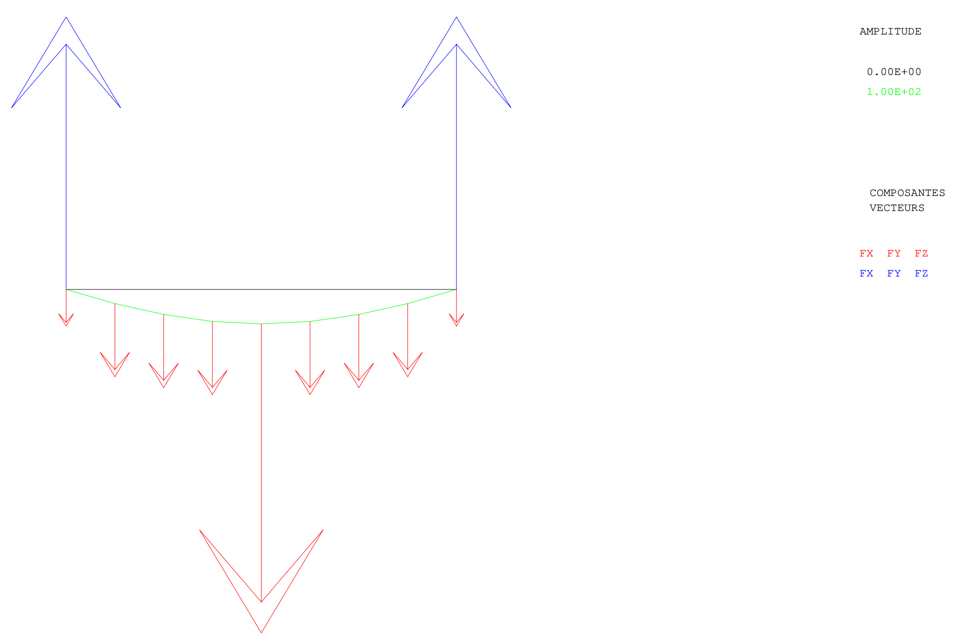
Test elas13 **Results**

RESULTS

	$Rot_A(Rad)$	$R_A(N)$	$U_Y(D)(m)$	$M_f(D)(N.m)$
Reference	$-3.045E-3$	$1.167E4$	$-4.233E3$	$3.384E4$
Castem	$-3.15250E-3$	11559	$-4.38085E3$	$3.3407E4$

CASTEM figures

ELAS13 : MESHING



ELAS13 : STRAIN, LOADING, REACTIONS

```
*          Test Elas13.dgibi: Jeux de données       *
*          ---------------------------------        *
*                                                   *
*****************************************************
* CAS TEST DU 91/06/13     PROVENANCE : TEST
SAUT PAGE;
*
*****************************************************
*                                                   *
*   TEST ELAS13                                     *
*                                                   *
*   POUTRE SUR SOL ELASTIQUE, EXTREMITES ARTICULEES  *
*                                                   *
*   Test V.P.C.S. numero SSLL16/89                  *
*                                                   *
*   Reference : M. Courtand et P. Lebelle, Formulaire *
*         du beton arme, T. 2, Paris, Eyrolles, 1976  *
*                                                   *
*   La structure est constituee d'une poutre articulee*
*   aux extremites et reposant sur un sol elastique.  *
*                                                   *
*      On se propose de calculer la rotation et la   *
*   reaction a une extremite, ainsi que la fleche et  *
*   le moment de flexion au centre de la poutre.      *
*                                                   *
*     Les valeurs trouvees sont ensuites comparees    *
*   aux valeurs de reference obtenues par solution    *
*   analytique.                                      *
*                                                   *
*****************************************************
*
OPTI ECHO 0;
*
OPTI DIME 3;
OPTI ELEM SEG2;
*
GRAPH = 'N';
*
*****************************************************
*                 maillage - geometrie              *
*****************************************************
*
L = PI * (10 ** .5) / 2.;
*
PA = (-.5 * L) 0. 0.;
PB = (0.5 * L) 0. 0.;
*
SUR1 = PA DROI 8 PB;
PD = SUR1 POIN PROC (0. 0. 0.);
*
SI (NEG GRAPH 'N');
  TITR 'ELAS13 : MAILLAGE';
  TRAC (0. 0. 1000.) QUAL SUR1;
FINSI;
*
*****************************************************
*              calcul de la rigidite                *
*****************************************************
*
MODL1 = MODL SUR1 MECANIQUE ELASTIQUE ISOTROPE POUT;
MATR1 = MATR MODL1 YOUN 2.E11 NU 0.3 TORS 1000.
                 SECT 1000 INRY 1.E-4 INRZ 1.E-4;
RIGI1 = RIGI MODL1 MATR1;
*
*****************************************************
*             conditions aux limites                *
*****************************************************
*
*  - Rotule en A et B,
*  - Appui continu elastique : sol de raideur lineique
*      constante : K = 840 000 N/m2 ,
*  - Conditions d'un probleme plan.
*
CL1 = BLOQ UX (PA ET PB);
CL2 = BLOQ UY (PA ET PB);
*
CL3 = BLOQ UZ SUR1;
CL4 = BLOQ RX SUR1;
*
NEL1 = NBEL SUR1;
K = 840000;
KNOEUD = K * L / (2. * NEL1);
I = 0;
CL5 = APPUI DEPL 0. SUR1;
REPETER BLOC2 NEL1;
  I = I + 1;
  POI1 = SUR1 POIN I;
  POI2 = SUR1 POIN (I + 1);
  APP1 = APPUI DEPL KNOEUD POI1;
  APP2 = APPUI DEPL KNOEUD POI2;
  CL5 = CL5 ET APP1 ET APP2;
FIN BLOC2;
*
RIGI1 = RIGI1 ET CL1 ET CL2 ET CL3 ET CL4 ET CL5;
*
*****************************************************
*                      chargement                   *
*****************************************************
*
*  - Force ponctuelle verticale de -10000 N au point D,
*  - Force lineique repartie verticale de -5000 N/m,
*  - Moments opposes aux extremites de 15000 N.m .
*
CHA1 = FORC (0. -10000. 0.) PD;
*
FTOTAL = -5000. * L;
FPOINT = FTOTAL / (2. * NEL1);
CHA2 = FORC FY 0. PA;
I = 0;
REPETER BLOC1 NEL1;
  I = I + 1;
  POI1 = SUR1 POIN I;
  POI2 = SUR1 POIN (I + 1);
  FOR1 = FORC FY FPOINT POI1;
  FOR2 = FORC FY FPOINT POI2;
  CHA2 = CHA2 ET FOR1 ET FOR2;
FIN BLOC1;
*
CHA3 = (MOMENT MZ -15000 PA) ET (MOMENT MZ 15000 PB);
*
CHAR1 = CHA1 ET CHA2 ET CHA3;
*
*****************************************************
*          resolution : champs de deplacements      *
*                       champ de contraintes        *
*****************************************************
*
DEPL1 = RESO RIGI1 CHAR1;
*
* Trace facultatif de la deformee
*
REAC1 = REAC RIGI1 DEPL1;
SI (NEG GRAPH 'N');
   TITR 'ELAS13 : DEFORMEE, CHARGEMENT, REACTIONS';
   VECT1 = VECT CHAR1 3.E-4 FX FY FZ ROUGE;
   VECT2 = VECT REAC1 3.E-4 FX FY FZ BLEU;
```

```
   DEF0 = DEFO SUR1 DEPL1 0.;
   DEF1 = DEFO SUR1 DEPL1 100. (VECT1 ET VECT2) VERT;
   TRAC (0. 0. 1000.) SUR1 (DEF0 ET DEF1);
FINSI;
*
SIGM1 = SIGM MODL1 MATR1 DEPL1;
SIGM2 = CHAN NOEUD MODL1 SIGM1;
SIGM3 = CHAN CHPO MODL1 SIGM2;
*
*******************************************************
*              extraction des resultats               *
*******************************************************
*
* Rotation du point A :
RZA1 = EXTR DEPL1 RZ PA;
RZA2 = -3.045E-3;
ERG1 = 100 * (ABS ((RZA2 - RZA1) / RZA2));
*
* Reaction au point A :
VA1 = EXTR REAC1 FY PA;
VA2 = 1.1674E4;
ERG2 = 100 * (ABS ((VA2 - VA1) / VA2));
*
* Fleche au point D :
UYD1 = EXTR DEPL1 UY PD;
UYD2 = -4.23326E-3;
ERG3 = 100 * (ABS ((UYD2 - UYD1) / UYD2));
*
* Moment de flexion au point D :
MD1 = EXTR SIGM3 MOMZ PD;
MD2 = 3.3840E4;
ERG4 = 100 * (ABS ((MD1 - MD2) / MD2));
*
*******************************************************
*               affichage des resultats               *
*******************************************************
*
MESS 'RESULTATS : TEST ELAS13';
MESS '-----------------------';
SAUT LIGN;
*
MESS ' Rotation du point A ';
MESS ' ------------------- ';
MESS '      Valeur theorique :' RZA2 'rad';
MESS '      Valeur calculee  :' RZA1 'rad';
MESS '                 Ecart :' ERG1 '%';
SAUT LIGN;
*
MESS ' Reaction au point A ';
MESS ' ------------------- ';
MESS '      Valeur theorique :' VA2 'N';
MESS '      Valeur calculee  :' VA1 'N';
MESS '                 Ecart :' ERG2 '%';
SAUT LIGN;
*
MESS ' Fleche au point D ';
MESS ' ----------------- ';
MESS '      Valeur theorique :' UYD2 'm';
MESS '      Valeur calculee  :' UYD1 'm';
MESS '                 Ecart :' ERG3 '%';
SAUT LIGN;
*
MESS ' Moment de flexion au point D ';
MESS ' ---------------------------- ';
MESS '      Valeur theorique :' MD2 'N.m';
MESS '      Valeur calculee  :' MD1 'N.m';
MESS '                 Ecart :' ERG4 '%';
SAUT LIGN;
*
*******************************************************
*                 code fonctionnement                 *
*******************************************************
*
ERGMAX = MAXI (PROG ERG1 ERG2 ERG3 ERG4);
SI (ERGMAX <EG 5.);
  ERRE 0;
SINON;
  ERRE 5;
FINSI;
*
SAUT 1 LIGN ;
*
TEMPS;
*
FIN;
*
```

Test elas13 **Comments**

- **Linear supports**

  APP1 = APPUI DEPL KNOEUD POI1;
  APP2 = APPUI DEPL KNOEUD POI2;

  The operator APPUI constructs the rigidity object associated with linear support. The keyword DEPL puts a support on every displacement components. KNOEUD is the value of the stiffness and POI1 or POI2 are the points where the support is imposed.

1.9 Test stru2 Description sheet

Test name stru2

Calculation type ELASTIC MECHANIC 2D, PLAN STRESS **Finite element type** QUA4

Topic
Embedded orthotropic square plate.

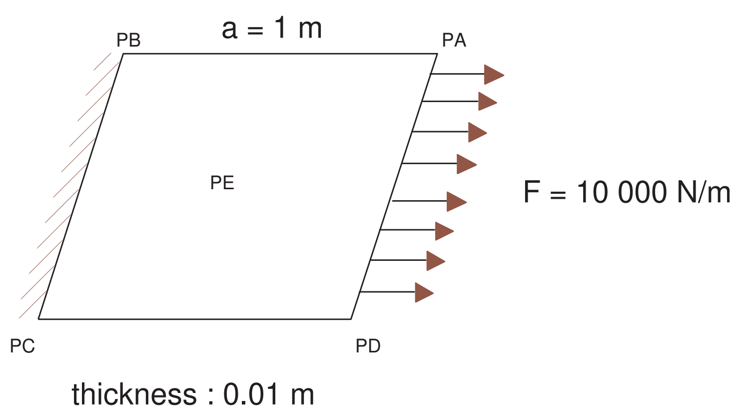
The plate is embedded at one side. A strength is applied to the opposite side, it is parallel to the plate. **Goal**

Find the horizontal and vertical displacements at the center of the plate (point PE) and at the two free angles (points PA and PD).

Reference CASTEM Comparison with six others finite element codes. **Version** 97' customer release

Model description

YOUNG1	=	$2.15737\ 10^{10}\ Pa$
YOUNG2	=	$7.377\ 10^{9}\ Pa$
YOUNG3	=	$7.377\ 10^{9}\ Pa$
NU12	=	0.31
NU23	=	0.31
NU13	=	0.31
G12	=	$2.319\ 10^{9} Pa$

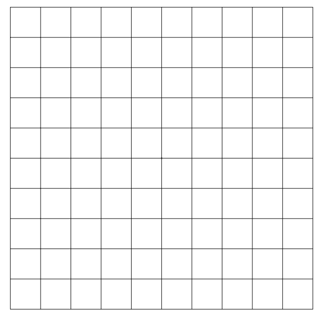


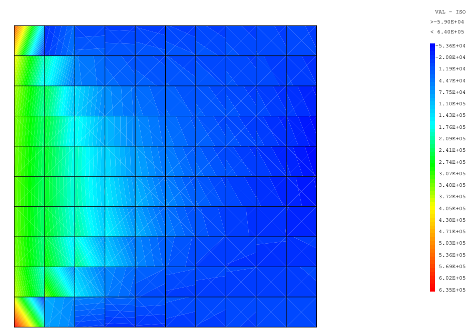
Test stru2 **Results**

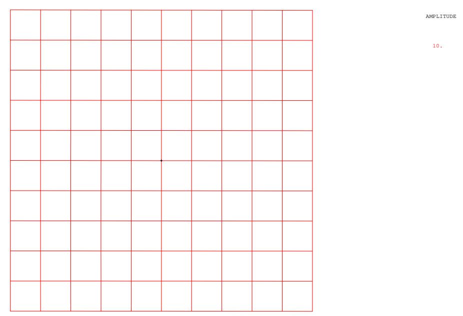
RESULTS

$(10^{-4}\ m)$	Displacement at PA		Displacement at PD		Displacement at PE	
	Horiz.	Verti.	Horiz.	Verti.	Horiz.	Verti.
Reference	1.09800	-1.30900	1.00300	-0.71340	0.47680	-0.49650
Castem	1.09961	-1.31276	0.99508	-0.71706	0.47668	-0.49812

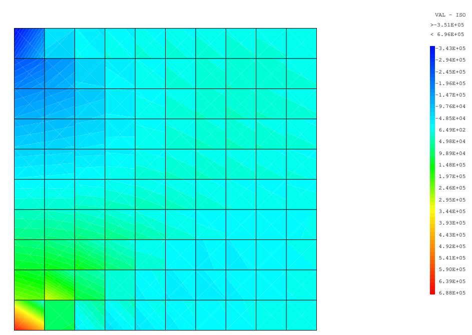
CASTEM FIGURES

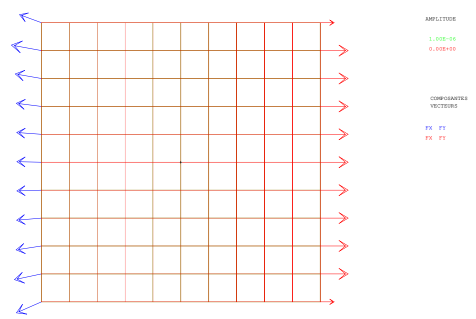

ORTHOTROPIC SQUARE PLATE


STRESS SMYY

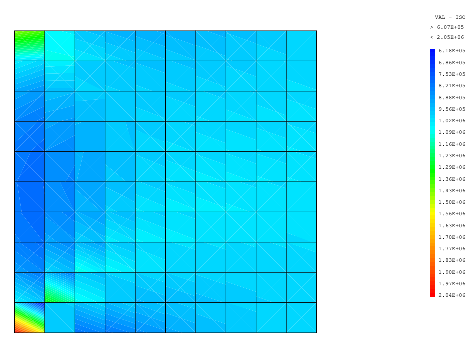


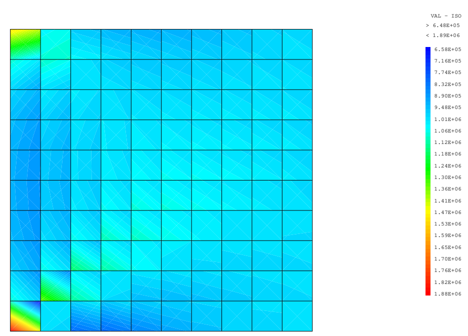
ORTHOTROPIC SQUARE PLATE : DEFORMED PLATE


STRESS SMXY



ORTHOTROPIC SQUARE PLATE : DEFORMED PLATE


VON-MISES STRESS



STRESS SMXX

```
*          Test Stru2.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION  ECHO 0;
*
GRAPH = 'N';
*
MESS ' *********************************************';
MESS ' *                                           *';
MESS ' * COMMISSION V.P.C.S.   TEST STRU2.DGIBI    *';
MESS ' *                       TEST SSLS 33/91     *';
MESS ' * GROUPE : STRUCTURE ASSEMBLEE              *';
MESS ' *         STATIQUE LINEAIRE                 *';
MESS ' *                                           *';
MESS ' * TEST : PLAQUE CARREE ORTHOTROPE           *';
MESS ' *        ENCASTREE                          *';
MESS ' *                                           *';
MESS ' * REMARQUE :  CAS TEST FAIT AVEC ELEMENTS   *';
MESS ' *             QUA4 EN CONTRAINTES PLANES    *';
MESS ' *                                           *';
MESS ' * LES VALEURS DE REFERENCES SONT LES        *';
MESS ' * MOYENNES DE RESULTATS OBTENUS PAR SIX     *';
MESS ' * CODES DIFFERENTS                          *';
MESS ' *                                           *';
MESS ' * TEST MODIFIE LE 3/07/92                   *';
MESS ' *********************************************';
*
OPTI TRAC BENS;
OPTION DIME 2 ELEM QUA4;
OPTION MODE PLAN CONT;
*
*-----------------------------------------------------*
*   GEOMETRIE                                         *
*-----------------------------------------------------*
*
PA =  0.5  0.5;
PB = -0.5  0.5;
PC = -0.5 -0.5;
PD =  0.5 -0.5;
PE =  0.   0.;
*
D1 = PA DROIT 10 PB;
D2 = PB DROIT 10 PC;
D3 = PC DROIT 10 PD;
D4 = PD DROIT 10 PA;
*
SUR1 = DALLAGE D1 D2 D3 D4 PLAN;
SUR = SUR1 ET PE;
*
ELIM SUR 0.001;
*
SI (NEG GRAPH 'N');
  TITRE 'PLAQUE CARREE ORTHOTROPE';
  TRACE SUR ;
FINSI ;
*
NBNOSUR = NBNO SUR1;
NBELSUR = NBEL SUR1;
*
*-----------------------------------------------------*
*        MODELE                                       *
*-----------------------------------------------------*
*
MOD = MODL SUR1 MECANIQUE ELASTIQUE ORTHOTROPE QUA4;
*
*-----------------------------------------------------*
*      CARACTERISTIQUES - MATERIAUX                   *
*-----------------------------------------------------*
*
DIR1 = (COS 30.) (SIN 30.);
*
MAT = MATR MOD DIRE DIR1 YG1 2.5737E10 YG2 7.377E9
YG3 7.377E9 NU12 0.31 NU23 0.31 NU13 0.31 G12 2.319E9;
*
*-----------------------------------------------------*
*          CONDITIONS AUX LIMITES  -  RIGIDITES       *
*-----------------------------------------------------*
*
CL = BLOQUE DEPL ROTA D2;
RI = RIGI MOD MAT ;
RIG = RI ET CL;
*
*-----------------------------------------------------*
*                     CHARGEMENTS                     *
*-----------------------------------------------------*
* IL S'AGIT D'UN CHARGEMENT LINEIQUE DE 1e4 NEWTONS/M
* SUR AD. L'EPAISSEUR EST DE 0.01M. ON A DONC UNE
* FORCE PAR UNITE D'EPAISSEUR DE 1E6 N/M

FTOT = 1.E6;
NBNOD4 = NBNO D4;
F5 = FTOT / ((2 * NBNOD4) - 2);
F50 = F5 0.;
I = 1;
NBI = NBNOD4 - 1;
CHA = FORC (0. 0.) (D4 POIN 1);
REPETER BLOC1 NBI;
  CHA1 = FORC F50 (D4 POIN I);
  CHA2 = FORC F50 (D4 POIN (I+1));
  CHA = CHA ET CHA1 ET CHA2;
  I = I + 1;
FIN BLOC1;

*-----------------------------------------------------*
*RESULTATS : DEPLACEMENTS,  CONTRAINTES               *
*-----------------------------------------------------*
*
DE = RESO RIG CHA;
*
GRAPH = 'O';
SI (NEG GRAPH 'N');
  TITR ' PLAQUE CARREE ORTHOTROPE : DEFORMEE .';
  DEF0 = DEFO SUR  DE 0.;
  DEF1 = DEFO SUR  DE 10. ROUGE;
  TRAC (DEF0 ET DEF1);
FINSI;
*
UA = EXTR DE UX PA; UA1 = 1.098e-4;
ERGUA = 100 * (ABS ((UA1 - UA) / UA1));
*
UD = EXTR DE UX PD; UD1 = 1.003e-4;
ERGUD = 100 * (ABS ((UD1 - UD) / UD1));
*
UE = EXTR DE UX PE; UE1 = 4.768E-5;
ERGUE = 100 * (ABS ((UE1 - UE) / UE1));
*
VA = EXTR DE UY PA; VA1 = -1.309E-4;
ERGVA = 100 * (ABS ((VA1 - VA) / VA1));
*
VD = EXTR DE UY PD; VD1 = -7.134E-5;
ERGVD = 100 * (ABS ((VD1 - VD) / VD1));
*
VE = EXTR DE UY PE; VE1 = -4.965E-5;
ERGVE = 100 * (ABS ((VE1 - VE) / VE1));
*
SAUT 2 LIGN;
```

```
MESS '    Resultats : test stru2.dgibi';
MESS '    ---------------------------';
SAUT 1 LIGN;
MESS ' Deplacements';
MESS ' ------------';
SAUT 1 LIGN;
*
MESS '   Point A :';
MESS '   ---------';
MESS '  Deplacement theorique horizontal :' UA1 'm';
MESS '  Deplacement calcule horizontal   :' UA 'm';
MESS '      Soit un ecart de :' ERGUA '%';
SAUT 1 LIGN;
MESS '  Deplacement theorique vertical   :' VA1 'm';
MESS '  Deplacement calcule vertical     :' VA 'm';
MESS '      Soit un ecart de :' ERGVA '%';
SAUT 1 LIGN;
*
MESS '   Point D :';
MESS '   ---------';
MESS '  Deplacement theorique horizontal :' UD1 'm';
MESS '  Deplacement calcule horizontal   :' UD 'm';
MESS '      Soit un ecart de :' ERGUD '%';
SAUT 1 LIGN;
MESS '  Deplacement theorique vertical   :' VD1 'm';
MESS '  Deplacement calcule vertical     :' VD 'm';
MESS '      Soit un ecart de :' ERGVD '%';
SAUT 1 LIGN;
*
MESS '   Point E :';
MESS '   ---------';
MESS '  Deplacement theorique horizontal :' UE1 'm';
MESS '  Deplacement calcule horizontal   :' UE 'm';
MESS '      Soit un ecart de :' ERGUE '%';
SAUT 1 LIGN;
MESS '  Deplacement theorique vertical   :' VE1 'm';
MESS '  Deplacement calcule vertical     :' VE 'm';
MESS '      Soit un ecart de :' ERGVE '%';
SAUT 1 LIGN;
*
*SAUT 1 LIGN;
*MESS ' Contraintes au point E';
*MESS ' ----------------------';
*SAUT 1 LIGN;
*
CHAM1 = SIGM MOD MAT DE;
CHAM2 = CHAN NOEUD MOD CHAM1;
CHPO1 = CHAN CHPO MOD CHAM2;
*
SMXXE = EXTR CHPO1 SMXX PE;
SMXXE = SMXXE / 1.E6;
SMXXE1 = 1.004;
ERGXX = 100 * (ABS ((SMXXE1 - SMXXE) / SMXXE1));
*
SMYYE = EXTR CHPO1 SMYY PE;
SMYYE = SMYYE / 1.E6;
SMXYE = EXTR CHPO1 SMXY PE;
SMXYE = SMXYE / 1.E6;
*
*MESS ' Contrainte theorique horizontale :' SMXXE1
*'MPa';
*MESS ' Contrainte calculee horizontale  :' SMXXE
*'MPa';
*MESS '      Soit un ecart de :' ERGXX '%';
SAUT 1 LIGN;
*MESS ' Contrainte calculee verticale    :' SMYYE
*'MPa';
SAUT 1 LIGN;
*MESS ' Contrainte calculee de cisaillement :' SMXYE
*'MPa';
SAUT 1 LIGN;
*
MESS ' Nombre de noeuds  :' NBNOSUR;
MESS ' Nombre d elements :' NBELSUR;
*
* UTILISATION DE LA PROCEDURE AFFICHE
*
SI (NEG GRAPH 'N');
  AFFICHE RIG CHA SUR 1.E-6 1.E-6;
FINSI;
*
SI (NEG GRAPH 'N');
  CHAM10 = EXCO SMXX CHAM1;
  TITRE 'CONTRAINTE SMXX';
  TRACE MOD CHAM10 SUR;
*
  CHAM11 = EXCO SMYY CHAM1;
  TITRE 'CONTRAINTE SMYY';
  TRACE MOD CHAM11 SUR;
*
  CHAM12 = EXCO SMXY CHAM1;
  TITRE 'CONTRAINTE SMXY';
  TRACE MOD CHAM12 SUR;
*
  CHAM13 = VMIS MOD CHAM1;
  TITRE 'CONTRAINTE DE VON-MISES';
  TRACE MOD CHAM13 SUR;
FINSI ;
*
*====================================================*
*                code fonctionnement                 *
*====================================================*
*
ERGMAX=MAXI (PROG ERGUA ERGUD ERGUE ERGVA ERGVD ERGVD);
*
* Il y a erreur si un des pourcentages ERG*
* est superieur a 5%
*
SAUT 2 LIGN;
SI (ERGMAX <EG 5.);
    ERRE  0 ;
SINO;
    ERRE  5 ;
FINSI;
*
SAUT 1 LIGN;
TEMPS;
SAUT 1 LIGN;
*
FIN;
*
```

Test stru2 **Comments**

1. **Construction of a field by element**

```
CHAM2 = CHAN NOEUD MOD CHAM1 ;
```

Depending on the data, the functions of this operator are different. Here the key word 'NOEUD' will bring about the CHAN operator to construct from a new field by element CHAM1 (MCHAML type) and from a model object MOD (MCHAML type), a field by element CHAM2 of the same type as that given on input, lying respectively on the nodes, centres of gravity, stiffness integration points, mass integration points, and element stress calculation points. For this purpose, a least squares method is used together with the shape functions of the element.

2. **Deformed shape display**

```
AFFICHE RIG CHA SUR 1.E-6 1.E-6 ;
```

This procedure displays, on the screen, the deformed shape of a structure subjected to a given loading.
RIG : structure stiffness matrix (RIGIDITE type)
CHA : structure loading (CHPOINT type)
SUR : structure mesh (MAILLAGE type)
1.E-6 : coefficient of amplification of the deformed shape (FLOTTANT type) and of the vectors representing the loading (FLOTTANT type)

1.10 Test stru4 Description sheet

Test name stru4

Calculation type ELASTIC MECHANIC, 3D **Finite element type** TUYAU

Topic
Embedded curved tube subjected to flexion

The curved tube is embedded at one side. A strengh is applied to the free end, it is parallel to the plan of the tube. **Goal** Find the horizontal and vertical displacement at the free end (point PB) and find the rotation of

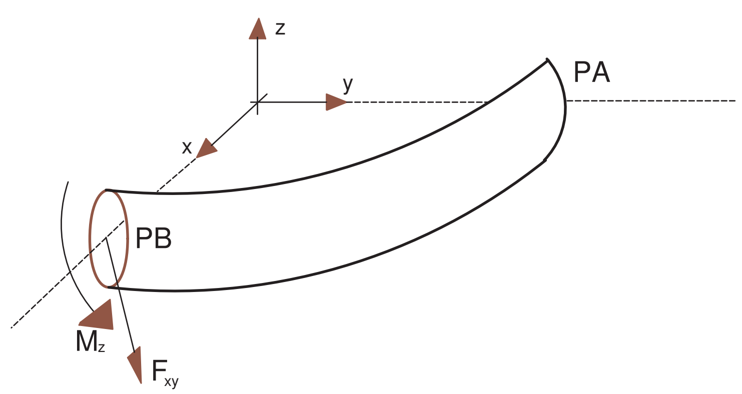
the tube.

Reference CASTEM Guide VPCS, Test SSLL 06/90
Solution analytique; JS Prezemieniecki;
Theory of matrix structural analysis
New York, MCGraw-Hill, 1968

Version 97' customer release

Model description

YOUNG	=	$2\ 10^{11}\ Pa$
NU	=	0.3
EPAISSEUR	=	$0.002\ m$
RAYON	=	$0.01\ m$

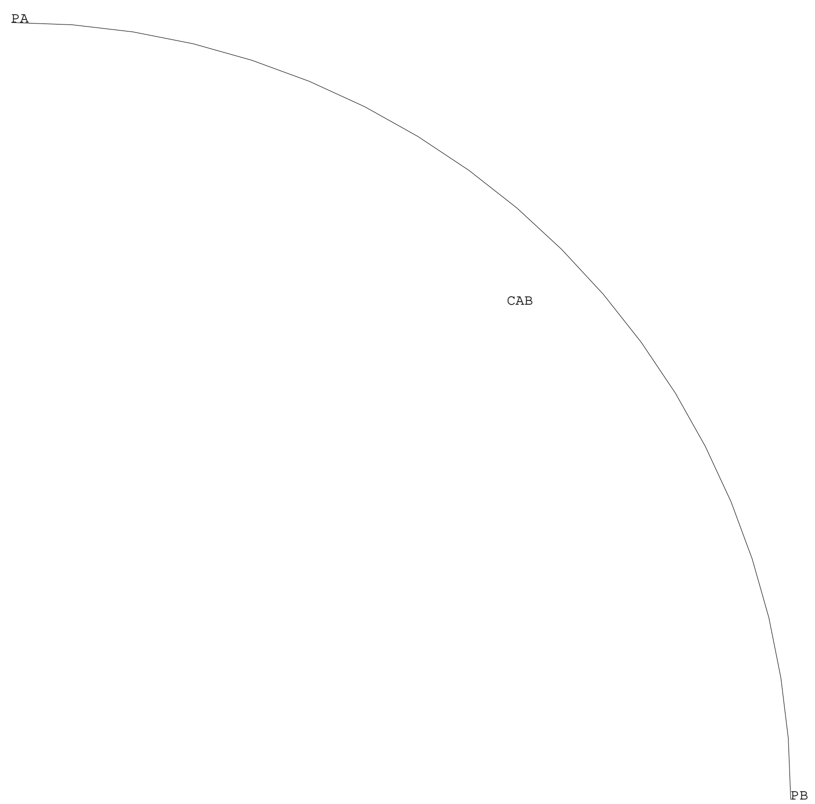


Test stru4 **Results**

RESULTS

	Horiz. displ. at PB	Vertical displ. at PB	Rotation
Reference	0.37908 m	0.24173 m	0.16539 rad
Castem	0.37871 m	0.24174 m	0.16533 rad

CASTEM FIGURES



AN EMBEDDED CURVED TUBE SUBJECTED TO FLEXION

```
*           Test Stru4.dgibi: Jeux de données        *
*           ---------------------------------        *
*                                                    *
******************************************************
*                                                    *
* TEST DE STATIQUE POUR L'ELEMENT TUYAU              *
*                                                    *
* Un tuyau encastre et courbe est soumis ades      *
* efforts de flexion dans son plan.                  *
*                                                    *
* Ref: Guide VPCS, Test SSLL 06/90                   *
*      Solution analytique; JS Prezemieniecki;       *
*      Theory of matrix structural analysis, New York,*
*      McGraw-Hill, 1968                             *
*                                                    *
* Test effectue le 25/08/92                         *
*                                                    *
******************************************************

GRAPH= 'N';
opti echo 0;
opti dime 3;
opti elem seg2;

************* GEOMETRIE *******************
P0 = 0. 0. 0.;
PA =  0. 3. 0.;
PB =  3. 0. 0.;
CAB = CERC 20 PA P0 PB;
OEIL = 0. 0. 1000.;

SI ( NEG GRAPH 'N');
TRAC OEIL CAB;
FINSI;

************* MODELE **********************

MODL1 = modl cab 'MECANIQUE' 'ELASTIQUE' 'TUYAU';
MAT1  = MATR MODL1 YOUN 2E11 NU 0.3 EPAI 0.002 RAYO
0.01 VECT OEIL;

************* CONDITIONS AUX LIMITES *******

BL1 = BLOQU DEPL ROTA PA;

************* RIGIDITE   *******************

RG1 = RIGI MODL1 MAT1;
RGTOT = RG1 ET BL1;

************* CHARGEMENT *******************

FOR1 = FORC FX 10. FY 5. PB;
MOM1 = MOME MZ 8 PB;
FORT = FOR1 ET MOM1;

************* CALCUL DEPLACEMENT *************

DEP = RESOU RGTOT FORT;
UBX = EXTR DEP PB UX;
UBY = EXTR DEP PB UY;
RBZ = EXTR DEP PB RZ;


************* CALCUL ANALYTIQUE **************

F1 = 10.; F2 = 5.; MZ = 8.; E= 2E11;
I = 4.637E-9; R = 3.;
UBX1 = (R**2)/4/E/I*((F1*PI*R)+(2*F2*R)+(4*MZ));
UBY1 = (R**2)/4/E/I*((2*F1*R)+(((3*PI)-(8.))*F2*R)+
(2*(PI-(2.))*MZ));
RBZ1 = R/4/E/I*((4*R*F1)+(2*(PI-(2.))*F2*R)+(2*PI*MZ));

************* COMPARAISON DES RESULTATS *****

MESS ' RESULTATS ';
MESS ' --------- ';

SAUT 2 LIGN ;
MESS ' DEPLACEMENT THEORIQUE AU POINT B SUIVANT X;'
UBX1 'M';
MESS ' DEPLACEMENT CALCULE   AU POINT B SUIVANT X;'
UBX  'M';
R1= 100*(ABS((UBX-UBX1)/UBX1));
MESS '     SOIT UN ECART DE       : '  R1  '%' ;
SAUT 2 LIGN ;
MESS ' DEPLACEMENT THEORIQUE AU POINT B SUIVANT Y;'
UBY1 'M';
MESS ' DEPLACEMENT CALCULE   AU POINT B SUIVANT Y;'
UBY  'M';
R2= 100*(ABS((UBY-UBY1)/UBY1));
MESS '     SOIT UN ECART DE       : '  R2  '%' ;
SAUT 2 LIGN ;
MESS ' ROTATION THEORIQUE AU POINT B;' RBZ1 'rad';
MESS ' ROTATION CALCULEE  AU POINT B;' RBZ 'rad';
R3= 100*(ABS((RBZ-RBZ1)/RBZ1));
MESS '     SOIT UN ECART DE       : '  R3  '%' ;


************* CODE DE FONCTIONNEMENT ********
LR = PROG R1 R2 R3;

ERMAX = MAXI LR;

SI ( ERMAX < 5);
   ERRE 0;
SINON
   ERRE 5;
FINSI;
FIN;
```

Test stru4 **Comments**

1. **GEOMETRICAL CHARACTERISTIC : TUYAU**

```
MODE1 = MODE CAB MECANIQUE ELASTIQUE TUYAU ;
MAT1 = MATE MODL1 YOUN 2E11 NU 0.3 EPAI 0.002
     RAYO 0.01 VECT OEIL ;
```

The TUYAU element is used in the modeling of parts of straight pipe or elbow which differ only in their radius of curvature. The pipe features are defined in the element local base, in the same way as the POUTRE element.
EPAI : thickness
RAYO : pipe external radius
VECT : key word for defining the local axis Oy. It must be followed by a POINT type object modeling an xOy vector.This is especially required when dealing with an elbow.

2. **APPLICATION OF A MOMENT**

```
MOM1 = MOME MZ 8 PB ;
```

The MOMENT operator creates a moment field resulting from the application of a moment represented either by the components of a vector, or by component values.
MZ : name of the components (MOT type)
8 : component values (FLOTTANT type) for PLAN CONT or PLAN DEFO
PB : geometrical support (MAILLAGE type)
MOM1 : moment field (CHPOINT type)

1.11 Test elas20 Description sheet

Test elas20 **Description sheet**

Test name elas20

Calculation type MECHANICS ELASTICITY 3D

Finite element type TUYAU

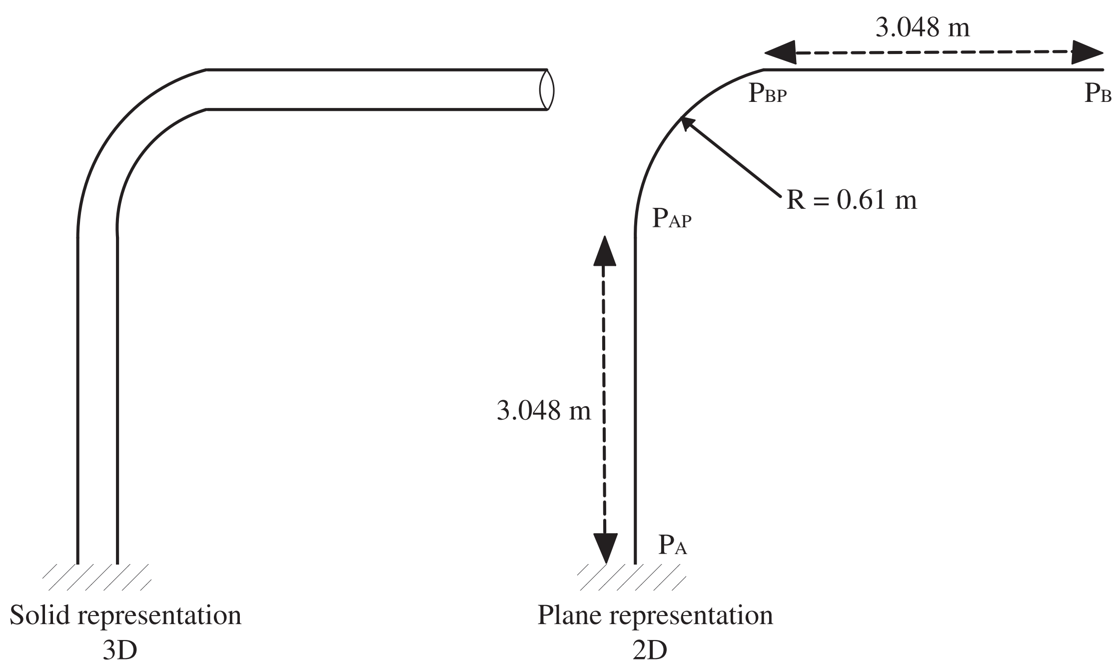
Topic Rubber-tubing subjected to gradient of pressure

The structure is composed by a one rubber-tubing : two linear parts separated by one crank part. The solid is modelised by a TUYAU finite element.

Goal Find the displacements at a points PB, PBP and PAP. The castem results are compared to the analytical results.

Reference CASTEM Test VPCS : SSLL08/90 : Modélisation des structures élastiques dans CASTEM 2000. J. S. Przemieniecki, theory of matrix structural analysis (New-York : Mcgraw-Hill Book Company, 1968) pp 155. **Version** 97' customer releas

Model description

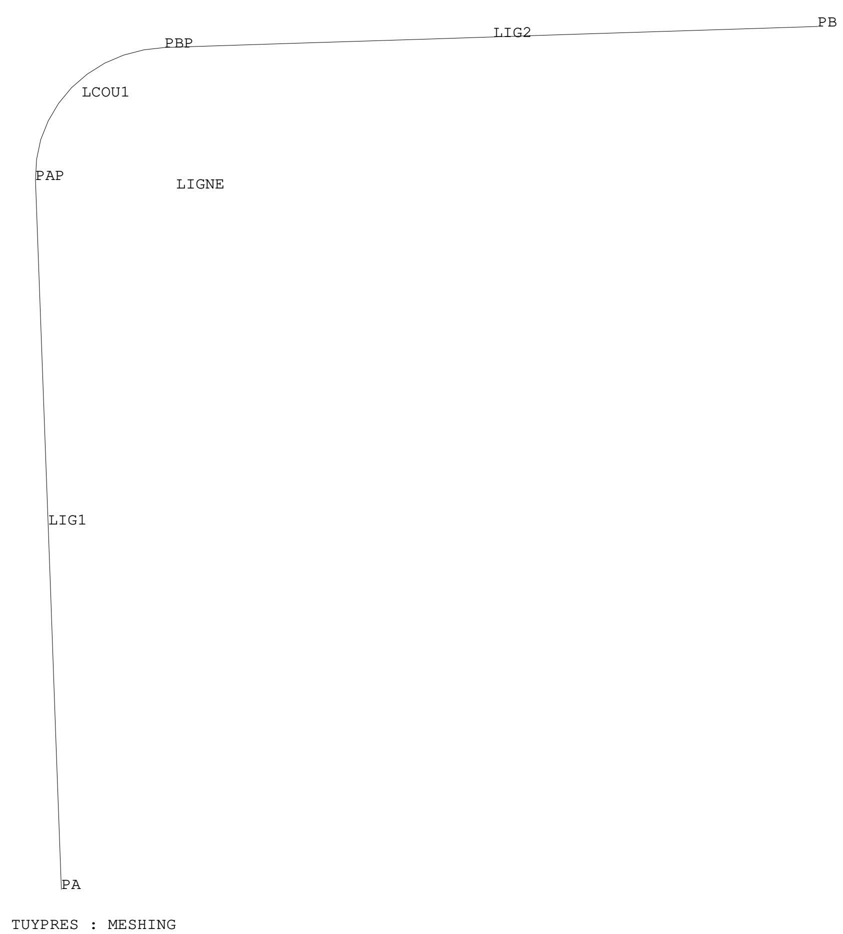


E	=	$210\ 10^{9}\ Pa$
NU	=	0.3
SECT	=	$1.13\ 10^{-4}\ m^{2}$

Test elas20 **Results RESULTS**

	U_x (PB)	U_y (PB)	U_x $(PBP)(m)$	U_y $(PBP)(m)$	U_y $(PAP)(m)$
Reference	$8.2200E-4$	$8.2200E-4$	$1.37E-4$	$8.22E-4$	$6.8500E-4$
Castem	$822692E-4$	$8.22692E-4$	$1.37544E-4$	$8.22692E-4$	$6.85148E-4$

CASTEM figures



```
*          Test Elas20.dgibi: Jeux de données     *
*          ---------------------------------      *
*                                                 *
SAUT PAGE ;
***************************************************
*                                                 *
* TEST TUYAU DROIT ET COUDE SOUS PRESSION         *
*                                                 *
* description :                                   *
* -----------                                     *
*                                                 *
*   Test en statique lineaire, chargement pression. *
*                                                 *
* La structure est modelisee en elements TUYAU :  *
*   - 2 parties droites de longueur 3.048m maillées *
*     par 5 éléments chacune.                     *
*   - 1 coude entre ces deux parties droites de   *
*     rayon de courbure égal à 0.61m, maillé par  *
*     10 éléments.                                *
*                                                 *
* Propriétés des matériaux :                      *
*   - Module d'Young        : YOUN1 = 2.e11       *
*   - Coefficient de Poisson : NU1   = 0.3        *
*                                                 *
* Chargement :                                    *
*   - Pression              : PRES1 = 15 Mpa      *
*                                                 *
* Conditions limites :                            *
*   - Encastrement a l'extrémité PA               *
*                                                 *
***************************************************
OPTI ECHO 0 ;
*
OPTI DIME 3 ELEM SEG2 ;
*
GRAPH = 'N' ;
*
***************************************************
*           GEOMETRIE : MAILLAGE                  *
***************************************************
*
OEIL = 0. 0. 10. ;
*
LDRO1 = 3.048 ;
RCO1  = 0.61 ;
LTOT1 = LDRO1 + RCO1 ;
*
PA  = 0. 0. 0. ;
PAP = 0. LDRO1 0. ;
PBP = RCO1 LTOT1 0. ;
PB  = LTOT1 LTOT1 0. ;
*
LIG1 = PA D 5 PAP ;
LIG2 = PBP D 5 PB ;
LCOU1 = CERC 10 PAP ( RCO1 LDRO1 0. ) PBP ;
*
LIGNE = LIG1 ET LCOU1 ET LIG2 ;
*
TASS LIGNE ;
ELIM LIGNE 1.E-5 ;
*
'SI' ( 'NEG' GRAPH 'N' ) ;
   'TITRE' 'TUYPRES : MAILLAGE' ;
   TRAC LIGNE OEIL QUAL ;
'FINSI' ;
*
***************************************************
*               MODELES                           *
***************************************************
*
OBLI1 = 'MODL' LIG1 MECANIQUE ELASTIQUE TUYAU ;
OBLI2 = 'MODL' LIG2 MECANIQUE ELASTIQUE TUYAU ;
OBCO1 = 'MODL' LCOU1 MECANIQUE ELASTIQUE TUYAU ;
OBLIGN = OBLI1 ET OBLI2 ET OBCO1 ;
*
***************************************************
*          VALEUR DE LA PRESSION                  *
***************************************************
*
PRES1 = 15.E6 ;
*
***************************************************
*               MATERIAUX                         *
***************************************************
*
MATLI1 = 'MATE' ( OBLI1 ET OBLI2 ) 'YOUN' 2.1E11 'NU' 0.3 ;
MATCO1 = 'MATE' OBCO1 'YOUN' 2.1E11 'NU' 0.3 ;
MATLIGN = MATLI1 ET MATCO1 ;
*
***************************************************
*             CARACTERISTIQUES                    *
***************************************************
*
CARLI1 = 'CARA' OBLI1 'EPAI' 26.187E-3 'RAYO' 203.2E-3
         'VECT' ( 0. 0. -1. ) 'PRES' PRES1 ;
CARLI2 = 'CARA' OBLI2 'EPAI' 26.187E-3 'RAYO' 203.2E-3
         'VECT' ( 0. 0. -1. ) 'PRES' PRES1 ;
CARCO1 = 'CARA' OBCO1 'EPAI' 26.187E-3 'RAYO' 203.2E-3
         'RACO' 0.61 'VECT' ( 0. 0. -1. ) 'PRES' PRES1;
CARLIGN = CARLI1 ET CARLI2 ET CARCO1 ;
*
CARTOT = MATLIGN ET CARLIGN ;
*
***************************************************
*          CONDITIONS LIMITES                     *
***************************************************
*
RIGBLOQ = 'BLOQ' 'DEPL' 'ROTA' PA ;
*
***************************************************
*          CALCUL DE LA RIGIDITE                  *
***************************************************
*
RIGLIGN = 'RIGI' OBLIGN CARTOT ;
*
RIGTOT = RIGLIGN ET RIGBLOQ ;
*
***************************************************
*             CHARGEMENT                          *
***************************************************
*
FPRES1 = 'PRES' 'TUYA' OBLIGN CARLIGN ;
*
***************************************************
*             RESOLUTION                          *
***************************************************
*
DEPL1 = 'RESO' RIGTOT FPRES1 ;
*
***************************************************
*        EXTRACTION DES RESULTATS                 *
***************************************************
*
VXB1 = 0.822E-3 ;
UXB1 = 'EXTR' DEPL1 'UX' PB ;
ERG1 = 100. * ( 'ABS' (( VXB1 - UXB1 ) / VXB1 )) ;
```

```
*
VYB1 = VXB1 ;
UYB1 = 'EXTR' DEPL1 'UY' PB ;
ERG2 = 100. * ( 'ABS' (( VYB1 - UYB1 ) / VYB1 )) ;
*
VXBP1 = 0.137E-3 ;
UXBP1 = 'EXTR' DEPL1 'UX' PBP ;
ERG3 = 100. * ( 'ABS' (( VXBP1 - UXBP1 ) / VXBP1 )) ;
*
VYBP1 = 0.822E-3 ;
UYBP1 = 'EXTR' DEPL1 'UY' PBP ;
ERG4 = 100. * ( 'ABS' (( VYBP1 - UYBP1 ) / VYBP1 )) ;
*
VYAP1 = 0.685E-3 ;
UYAP1 = 'EXTR' DEPL1 'UY' PAP ;
ERG5 = 100. * ( 'ABS' (( VYAP1 - UYAP1 ) / VYAP1 )) ;
*
*****************************************************
*         AFFICHAGE DES RESULTATS                    *
*****************************************************
*
'MESS' 'RESULTATS : TEST TUYPRES' ;
'MESS' '------------------------' ;
SAUT LIGN ;
*
'MESS' ' deplacement ux (m) du point PB' ;
'MESS' ' ------------------------------' ;
'MESS' '        Valeur theorique : ' VXB1 'm' ;
'MESS' '        Valeur calculee  : ' UXB1 'm' ;
'MESS' '                  Ecart  : ' ERG1 '%' ;
SAUT LIGN ;
*
'MESS' ' deplacement uy (m) du point PB' ;
'MESS' ' ------------------------------' ;
'MESS' '        Valeur theorique : ' VYB1 'm' ;
'MESS' '        Valeur calculee  : ' UYB1 'm' ;
'MESS' '                  Ecart  : ' ERG2 '%' ;
SAUT LIGN ;
*
'MESS' ' deplacement ux (m) du point PBP' ;
'MESS' ' -------------------------------' ;
'MESS' '        Valeur theorique : ' VXBP1 'm' ;
'MESS' '        Valeur calculee  : ' UXBP1 'm' ;
'MESS' '                  Ecart  : ' ERG3 '%' ;
SAUT LIGN ;
*
'MESS' ' deplacement uy (m) du point PBP' ;
'MESS' ' -------------------------------' ;
'MESS' '        Valeur theorique : ' VYBP1 'm' ;
'MESS' '        Valeur calculee  : ' UYBP1 'm' ;
'MESS' '                  Ecart  : ' ERG4 '%' ;
SAUT LIGN ;
*
'MESS' ' deplacement uy (m) du point PAP' ;
'MESS' ' -------------------------------' ;
'MESS' '        Valeur theorique : ' VYAP1 'm' ;
'MESS' '        Valeur calculee  : ' UYAP1 'm' ;
'MESS' '                  Ecart  : ' ERG5 '%' ;
*
ERGMAX = 'MAXI' ( 'PROG' ERG1 ERG2 ERG3 ERG4 ERG5 ) ;
*
'SI' ( ERGMAX <EG 5.) ;
  ERRE 0 ;
'SINON' ;
  ERRE 5 ;
'FINSI' ;
SAUT LIGN ;
*
TEMPS ;
*
FIN ;
```

Test elas20 **Comments**

- **Internal pressure in a TUYAU element**

  CARLI1 = 'CARA' OBLI1 'EPAI' 26.187E-3 'RAYO' 203.2E-3 'VECT' (0. 0. -1.) 'PRES' PRES1 ;
  CARLI2 = 'CARA' OBLI2 'EPAI' 26.187E-3 'RAYO' 203.2E-3 'VECT' (0. 0. -1.) 'PRES' PRES1 ;
  CARCO1 = 'CARA' OBCO1 'EPAI' 26.187E-3 'RAYO' 203.2E-3 'RACO' 0.6 'VECT' (0. 0. -1.)
  'PRES' PRES1 ;

  The geometrical characteristics of TUYAU elements can be completed by giving the internal pressure (with keyword PRES) and the radius of curvature if it is an elbow (with keyword RACO).

1.12 Test testICQ4 Description sheet

Test name testICQ4

Calculation type ISOTROPIC ELASTIC MECHANIC, PLAN DEFO **Finite element type** ICQ4

Topic Square plate sujected to hydrostatic and deviatoric loads.

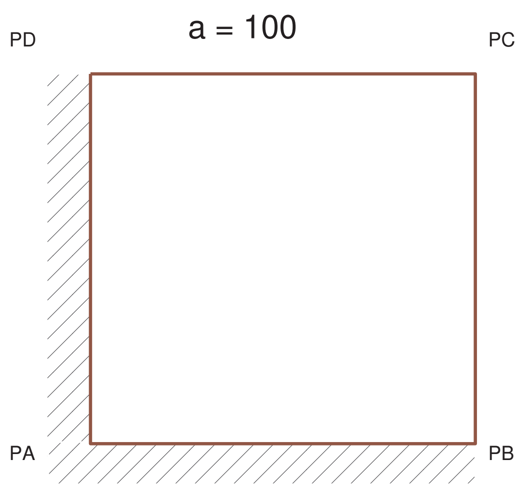
A square plate is embedded at two adjacent sides. Deviatoric and hydrostatic loads are applied to each side.
Goal Test the ICQ4-incompressible elements.

Reference CASTEM Rapport DMT sur les elements ICQ4.

Version 97' customer release

Model description

YOUNG	=	$2\ 10^{11}\ Pa$
NU	=	0.499
COEFF. HYDROSTATIC	:	$ALPHA = 1.$
COEFF. DEVIATORIC	:	$BETA = 1.E-3$



```
*          Test TestICQ4.dgibi: Jeux de données      *
*          ------------------------------------      *
*                                                    *
*
* FICHIER GIBIANE POUR TESTER LES ELEMENTS
* INCOMPRESSIBLES ICQ4
*
* Maillage d'une plaque de forme carrée avec les
* éléments ICQ4.
*
* Cas déformation plane.
*
* On soumet la plaque à un champ de contrainte
* hydrostatique (contribution: alpha=1) et à un champ
* de contrainte déviatorique (contribution: beta=0.001)
*
* Les résultats théoriques sont déterminés dans le cas
* incompressible (coefficient de Poisson proche de 0.5)
*
* cf rapport DMT sur ces elements
*
*
*-----MAILLAGE-----------
*
opti dime 2 mode plan defo elem qua4 ;
*
p1 = 0. 0. ;
p2 = 100. 0. ;
p3 = 100. 100. ;
p4 = 0. 100. ;
*
li1 = d 10 p1 p2 ;
li2 = d 10 p2 p3 ;
li3 = d 10 p3 p4 ;
li4 = d 10 p4 p1 ;
*
sur1 = dall li1 li2 li3 li4 'PLAN' ;
*
*
*-----MODELE MECANIQUE------------
*
* Valeur du coefficient de Poisson
*
nu1 = 0.499 ;
*
mod1 = modele sur1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       icq4 ;
mat1 = mate mod1 'YOUNG' 2.e11 'NU' nu1 ;
rigi1 = rigi mod1 mat1 ;
*
*
*-----CONDITIONS AUX LIMITES------------
*
bl1 = bloq li1 'UY' ;
bl2 = bloq li4 'UY' ;
bl3 = bloq p1 'UX' ;
*
*
*----------------CHARGEMENT-----------------
*
* Coefficients de contribution de chargement
* (alpha-->hydrostatique)
* (beta-->déviatorique)
*
alpha = 1. ;
beta = 1.e-3 ;
*
*-----Chargement déviatorique------------
*
*-----sur li1-----
*
chpx1 = coor 1 li1 ;
chpresy1 = pres 'MASS' mod1 (chpx1 * (-2.)*1.e5) ;
chpres1 = beta * chpresy1 ;
vecfor1 = vect chpres1 1.E-7 'FX' 'FY' rouge ;
*
*-----sur li2-----
*
chpy2 = coor 2 li2 ;
chpresx2 = pres 'MASS' mod1 (200.*1.e5) li2 ;
chpresy2 = pres 'MASS' mod1 (chpy2 * (-2.)*1.e5) ;
chforx2 = chpresx2 ;
chfory2 = exco 'FX' chpresy2 'FY';
chpres2 = beta * ( chforx2 et chfory2 ) ;
vecfor2 = vect chpres2 'FX' 'FY' 1.E-7 rouge ;
*
*-----sur li3-----
*
chpx3 = coor 1 li3 ;
chpresx3 = pres 'MASS' mod1 (-200.*1.e5) li3 ;
chpresy3 = pres 'MASS' mod1 (chpx3 * (-2.)*1.e5) ;
chforx3 = exco 'FY' chpresx3 'FX';
chfory3 = chpresy3 ;
chpres3 = beta * ( chforx3 et chfory3 ) ;
vecfor3 = vect chpres3 'FX' 'FY' 1.E-7 rouge ;
*
*-----sur li4-------
*
chpy4 = coor 2 li4 ;
chpresy4 = pres 'MASS' mod1 (chpy4 * (-2.)*1.e5) ;
chfory4 = exco 'FX' chpresy4 'FY';
chpres4 =  beta * chfory4 ;
vecfor4 = vect chpres4 1.E-7 'FX' 'FY' rouge ;
*
*
*-----Chargement hydrostatique------------
*
*-----sur li1-----
*
presy1 = pres 'MASS' mod1 (1.e5) li1 ;
pres1 = alpha * presy1 ;
vfor1 = vect pres1 1.E-5 'FX' 'FY' vert ;
*
*-----sur li2-----
*
presx2 = pres 'MASS' mod1 (1.e5) li2 ;
pres2 = alpha * presx2 ;
vfor2 = vect pres2 'FX' 'FY' 1.E-5 vert ;
*
*-----sur li3-----
*
presy3 = pres 'MASS' mod1 (1.e5) li3 ;
pres3 = alpha * presy3 ;
vfor3 = vect pres3 'FX' 'FY' 1.E-5 vert ;
*
*-----sur li4-------
*
presx4 = pres 'MASS' mod1 (1.e5) li4 ;
pres4 = alpha * presx4 ;
vfor4 = vect pres4 1.E-5 'FX' 'FY' vert ;
*
*
*---------CALCUL DES DEPLACEMENTS --------------------
*
*
* Solution analytique au point P3: uyan.
```

```
*
x3 = coor 1 p3 ;
y3 = coor 2 p3 ;
mu = 1.e11 / ( 1 + nu1 ) ;
uyan = ( 2 * x3 * y3 ) / ( 2 * mu ) * 1.e5 * beta ;
*
* Solution Castem au point P3: uycas.
*
deptot = reso (rigi1 et bl1 et bl2 et bl3)
(chpres2 et chpres3 et chpres4 et chpres1
et pres2 et pres3 et pres4 et pres1) ;
uycas = extr deptot 'UY' p3 ;
*
* Calcul de l'erreur sur le déplacement.
*
erry = abs (( uyan - uycas ) / ( uyan )) * 100 ;
*
* Test de précision sur le calcul du déplacement uy
* (<1%).
*
si ( erry >eg 1. ) ;
     erreur 5 ;
finsi ;
*
*
*-----------CALCUL CONTRAINTES-------------------------
*
*
* Solution analytique: pression = alpha*1.E5.
*
* Solution Castem : pression maximale = maxpres1.
*
sig1 = sigma mod1 deptot mat1 ;
sxx = exco 'SMXX' sig1 'P' ;
syy = exco 'SMYY' sig1 'P' ;
szz = exco 'SMZZ' sig1 'P' ;
sp = (sxx + syy + szz) / 3. ;
maxpres1 = maxi sp ;
*
* Calcul de l'erreur sur la pression.
*
errpress = abs ((maxpres1 + (alpha * 1.E5)) /
           (alpha * 1.E5)) * 100. ;
*
* Test de précision sur le calcul de la pression (<1%).
*
si ( errpress >eg 1. ) ;
     erreur 5 ;
finsi ;
*
*
fin;
```

Test testICQ4 **Comments**

1. ICQ4 INCOMPRESSIBLE ELEMENT

```
OPTI DIME 2 MODE PLAN DEFO ELEM QUA4 ;
MOD1 = MODE SUR1 MECANIQUE ELASTIQUE ISOTROPE ICQ4 ;
```

   The incompressible quadrangle with four nodes ICQ4 is a finite element of the QUA4 geometrical support in MECANIQUE.

2. DEVIATORIC LOAD

```
BETA = 1.E-3 ;
*-----ON LI1-----
CHPX1 = COOR 1 LI1 ;
CHPRESY1 = PRES MASS MOD1 (CHPX1 * (-2.)*1.E5) ;
CHPRES1 = BETA * CHPRESY1 ;
VECFOR1 = VECT CHPRES1 1.E-7  FX   FY  ROUG ;
```

   The COOR operator enables the user to recover the Nith coordinate (here the first, then UX) of the MAILLAGE (LT1) type object.
   The PRES operator calculates the nodal forces which are equivalent to a pressure applied on an object. Here this operator is connected by key words MASS reffering to the type of element on which is applied. MOD1 is the object on which the pressure is appiled (MMODEL type). The pressure (FLOTTANT type) (-2)*1.E-5 is negative because it is carried by the negative normal to the element.
   The VECT operator constructs a VECTEUR type object (VECTFOR1) from the components of a vector field (CHPRES1), and 1.E-7 is the coefficient of amplification (FLOTTANT type). The names of the components of the field associated with the directions Ox, Oy (MOT type) are FX and FY.

3. HYDROSTATIC LOAD

```
ALPHA = 1.
PRESX2 = PRES MASS MOD1 (1.E5) LI2 ;
PRES2 = ALPHA * PRESX2 ;
VFOR2 = VECT PRES2  FX   FY  1.E-5 VERT ;
```

1.13 Test vsur1 Description sheet

Test name vsur1

Calculation type ISOTROPIC ELASTIC MECHANIC **Finite element type** COQ2

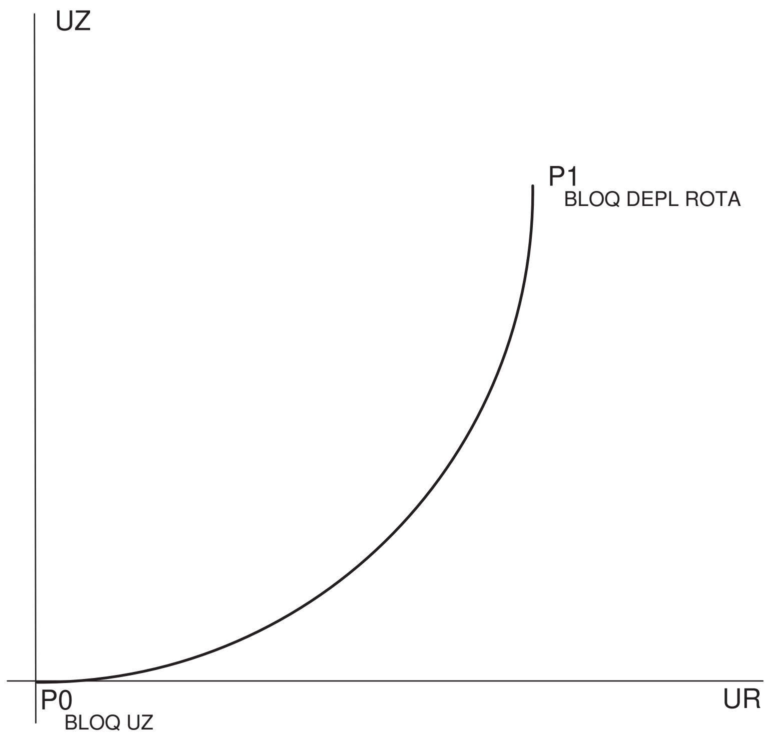
Topic VSUR operator test.

A circular structure is subjected to a surface vector field, a normal vector field and a field of jacobian.

Goal Test the VSUR operator with COQ2 elements.

Version 97' customer release

Model description



```
*          Test Vsur1.dgibi: Jeux de données        *
*          --------------------------------         *
*                                                   *
*****************************************************
*   vsur1.dgibi : Test de l'opérateur VSUR en coq2    *
*****************************************************
opti echo 0 dime 2 elem seg2 mode axis;
*
*  Définition de la géométrie
*
p0 = 0. 0.;
p1 = 10. 10.;
pc = 0. 10.;
obj1 = cerc 10 p0 pc p1;
cl1 = bloq 'DEPL' 'ROTA' p1;
cl2 = bloq 'UZ' p0;
*
*  Création du modèle
*
mod = modl obj1  mecanique elastique isotrope coq2;
*
*  Comparaison des champs obtenus par VSUR
*
* che1 : champ de vecteurs surfaciques
che1 = VSUR mod;
* chn1 : champ de normaes
chn1 = VSUR mod 'NORM';
* chj1 : champ des jacobiens
chj1 = JACO mod;
cnr1 = exco 'VR' che1 'SCAL';
cnz1 = exco 'VZ' che1 'SCAL';
* champ des normes
cnn1 = (cnr1*cnr1) + (cnz1*cnz1);
cjn1 = chj1*chj1;
cze1 = cnn1 - cjn1;
*list cze1;
cpz1 = chan 'CHPO' mod cze1;
res1 = maxi cpz1 'ABS';
*list res1;
cnr1 = exco 'VR' chn1 'SCAL';
cnz1 = exco 'VZ' chn1 'SCAL';
cnn1 = (cnr1*cnr1) + (cnz1*cnz1);
*list cnn1;
cpn1 = chan 'CHPO' mod cnn1;
res2 = mini cpn1 'ABS';
res3 = maxi cpn1 'ABS';
*list res2;
*list res3;
EC1 = RES3 - RES2;
SI ((res1 < 1.e-6) ET (ABS(res2 - 1.) < 1.E-6) ET (EC1 < 1.E-6)) ;
  MESS 'OPERATEUR <VSUR> ERR  0';
  ERRE  0;
SINON;
  MESS 'OPERATEUR <VSUR> ERR  5';
  ERRE  5;
FINSI;
fin;
```

Test vsur1 **Comments**

1. VSUR OPERATOR

```
CHE1 = VSUR MOD ;
CHN1 = VSUR MOD NORM
```

The VSUR operator enables the user to calculate the surface vectors associated with the integration points of the shell elements, for the other types of elements the vector field is empty. If the NORM key-word is not specified, the norm of the vectors is equal to the jacobians at the considered integration point, otherwise the vector field is the normal vector field at the shell elements. Here MOD is the model object (MMODEL type) and CHE1 and CHN1 are the generated objects (MCHAML type, SCALAIRE subtype).

2. JACOBIEN OPERATOR

```
CHJ1 = JACO MOD ;
```

The JACOBIEN operator enables the user to calculate the absolute value of Jacobians at the integration points of the elements of the model. Here MOD is the model object (MMODEL type) and CHJ1 is the generated object (MCHAML type, SCALAIRE subtype).

Chapter 2

BUCKLING

2.1 Test flam1 Description sheet

Test name flam1

Calculation type MECHANICS BUCKLING. 2D PLANE STRESS

Finite element type COQ2

Topic
Eulerian buckling of a beam clamped at one end.

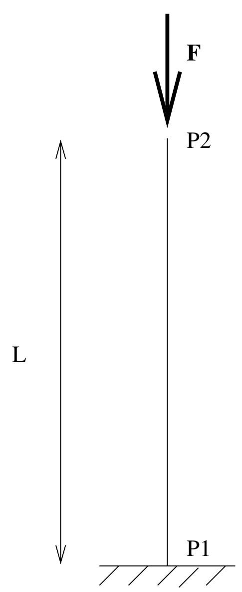
The beam is clamped at one end. A force, parallel to the beam, is applied to the free end.

Goal Find the strength multiplier M_s.

Reference Analytical solution.

Version 97' customer version

Model description



$$L = 10mm \quad Th = 1mm \quad F = 1N$$

$$E = 20000MPa \quad \nu = 0.3$$

Test flam1

Results

	M_s
Analytic	41.123
Castem	41.123

Results

CASTEM figures

```
*          Test Flam1.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0;
GRAPH='N';
SAUT PAGE ;
*
*******************************************************
*
*  TEST FLAM1
*
*          FLAMBAGE EULERIEN D'UNE POUTRE
*            ENCASTREE A UNE EXTREMITE
*
* Dans cet exemple on se propose d'étudier le flambage
* d'une poutre encastrée à une de ses extrémités et
* soumise à une force parallele à la poutre et
* appliquée à l'extrémité libre de la poutre.
*
* Le flambage est visualisé en fin de calcul et le
* multiplicateur de chargement calculé est comparé à
* celui obtenu par la théorie (41.123).
*
*******************************************************
TITRE ' FLAMBAGE D UNE POUTRE ENCASTREE ' ;
OPTION DIME 2 ELEM SEG2  MODE PLAN CONT ;


TEMPS ;
*
*------------- DEFINITION DU MAILLAGE -----------------
*
DENSITE  1. ;
P1 = 50. 0. ;
P2 = 50. 10. ;
L1 = P1 D P2 ;
*
SI (NEG GRAPH 'N');
TRAC L1 ;
FINSI;
*
*--------- CREATION DU MODELE ET DE MMODEL ------------
*
MOD1 = MODL L1 MECANIQUE ELASTIQUE COQ2;

*
* -------- DEFINITION DES CARACTERISTIQUES ------------
* ----------- MATERIELLES ET GEOMETRIQUES  ------------
*
MAT1   = MATR MOD1 YOUN   20000.    NU 0.3     ;
CAR1   = CARB MOD1 EPAI 1. ;
MAT1=MAT1 ET CAR1;
*
*--------- CALCUL DES RIGIDITES ELEMENTAIRES ----------
*----------- ET DEFINITIONS DES BLOCAGES --------------
*
RIG1   = RIGI  MOD1 MAT1 ;
BL1    = BLOQUE DEPL ROTA P1 ;
RIG11  = RIG1 ET BL1;
*
*------------- DEFINITION DU CHARGEMENT ---------------
*
FE1 = FORCE ( 0 -1. ) P2 ;
*
*------- RESOLUTION ET CALCUL DES CONTRAINTES ---------
*
DEP1   = RESO  RIG11    FE1 ;
SIG1 = SIGM MOD1 MAT1  DEP1 ;
*
*--- CALCUL DE LA MATRICE DES CONTRAINTES INITIALES ---
*
KSI1   = KSIGMA  MOD1 SIG1 CAR1 'FLAM' ;
*
*----------- INITIALISATION DE LA TABLE ---------------
*--------- EN ENTREE DE LA PROCEDURE FLAMBAGE ---------
*
ETAB   = TABLE    ;
ETAB.'CLIM' = BL1 ;
*ETAB.'RIGI' = RIG1 ;
*ETAB.'KSIG' = KSI1 ;
ETAB.'SIG1'=SIG1;
ETAB.'MATE'=MAT1;
ETAB.'LAM1' = 0.001 ;
ETAB.'LAM2' =  100. ;
ETAB.'NMOD' = 1   ;
ETAB.'OBJM'=MOD1;
*
*--------- APPEL A LA PROCEDURE DE FLAMBAGE -----------
*
SAUT PAGE ;
STAB = FLAMBAGE ETAB ;
*
*------------ RECUPERATION DES RESULTATS --------------
*
LA1  =  STAB . 1 . LAMB ;
SAUT PAGE ;
*
*------------ DESSIN DU MODE DE FLAMBAGE --------------
*
MM1 = STAB . 1 . DEPL ;
DE  = DEFO MM1 L1 1. ROUGE ;
DA  = DEFO MM1 L1 0. BLEU  ;

SI (NEG GRAPH 'N');
TRAC ( DE ET DA ) ;
FINSI;

*
MESS ' ' ;
MESS ' ' ;
MESS ' MULTIPLICATEUR DU CHARGEMENT ' ;
MESS '                              ' ;
MESS '   SOLUTION ANALYTIQUE  :   41.123 ' ;
MESS '   SOLUTION CALCULEE    : ' LA1 ;
*
TEMPS ;
*------------ CODE DE BON FONCTIONNEMENT --------------
ERR=100*(ABS(41.123 - LA1)/41.123);
SI (ERR < 5);
ERRE 0;
SINON;
ERRE 5;
FINSI;

FIN;
```

Test flam1 **Comments**

1. **Computation of initial stresses matrix**

   KSI1 = KSIGMA MOD1 SIG1 CAR1 'FLAM' ;

   It is necessary to compute the initial stresses matrix in order to make a buckling analysis. The operator KSIGMA builds the geometrical rigidity matrix joined to a field of stress. The keyword FLAM is necessary to specify that the matrix will be used for a buckling analysis, the created object is also a mass matrix (without the keyword FLAM, the object would be a rigidity matrix).

2. **Initialisation of input table of the 'flambage' subroutine**

   ETAB = TABLE ; ETAB.'CLIM' = BL1 ;
   *ETAB.'RIGI' = RIG1 ; ETAB.'KSIG' = KSI1 ;
   ETAB.'SIG1' = SIG1; ETAB.'MATE' = MAT1;
   ETAB.'LAM1' = 0.001 ; ETAB.'LAM2' = 100. ;
   ETAB.'NMOD' = 1 ; ETAB.'OBJM'= MOD1;

   ETAB is a table which contains the input data for the using of 'flambage' subroutine. This table is defined by different indices. The indices CLIM, LAM1, LAM2, NMOD and OBJM are necessary. The both other indices are either SIG1 and MATE or RIGI and KSIG.

   - CLIM : matrix joined to the clampings
   - LAM1 : minimal multiplier coefficient
   - LAM2 : maximal multiplier coefficient
   - NMOD : number of eigen shapes
   - OBJM : model object
   - SIG1 : yield of stress changing with the strength multiplier
   - MATE : material properties
   - RIGI : rigidity matrix
   - KSIG : geometrical rigidity matrix

3. **'Flambage' subroutine**

   STAB = FLAMBAGE ETAB ;
   LA1 = STAB . 1 . LAMB ;
   MM1 = STAB . 1 . DEPL ;

   STAB is a table which contains the output data given by the 'flambage' subroutine. The index of STAB square with every mode shape number so there are as many tables as number of computed moden shapes.
   Each table joined to a mode shape contains four objects which are indexed by :

   - LAMB : strength multiplier coefficient
   - DEPL : deformed and normed shape
   - MODN : number of Fourier harmonic
   - MODM : number of azimuth mode shape

2.2 Test four1 Description sheet

Test name four1

Calculation type MECHANICS BUCKLING. 2D FOURIER

Finite element type COQ2

Topic
Endless cylinder under external pressure.

This test simulates an endless cylinder subjected to an external pressure.

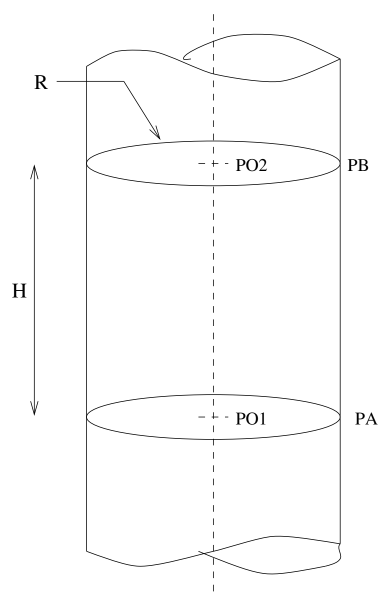
The first natural frequency of the structure's second mode shape is determined by a Fourier analysis. This result is used to make a buckling analysis.

Goal Find the strength multipliers M_s1 and M_s2.

Reference Analytic solution.

Version 97' customer release

Model description



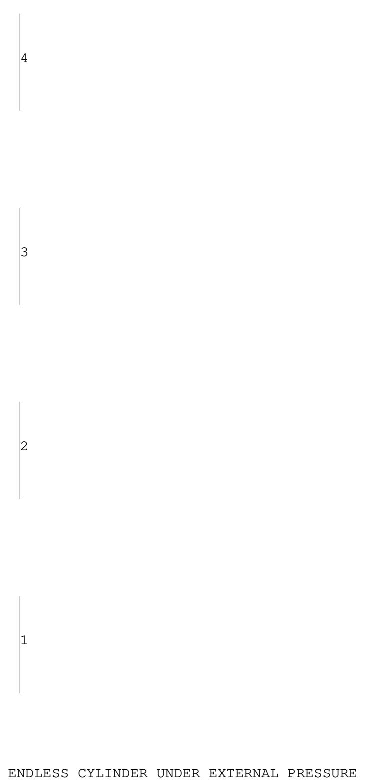
$$
\begin{array}{llll}
L & = 100mm & & \\
R & = 1000mm & E & = 20000MPa \\
Th & = 1mm & \nu & = 0 \\
P & = 1MPa & &
\end{array}
$$

Test four1

Results

	M_s1	M_s2
Analytic	$6.66.10^{-6}$	5.10^{-6}
Castem	$6.659.10^{-6}$	$4.995.10^{-6}$

CASTEM figures



Results

```
*          Test Four1.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0;
GRAPH='N';
SAUT PAGE ;
*******************************************************
*  TEST FOUR1
*
*  CYLINDRE INFINI SOUS PRESSION EXTERNE(NU=0)
*
* Soit un cylindre infini soumis à une pression externe
*
* Par une analyse de Fourier, on se propose de
* déterminer la première fréquence propre du deuxième
* mode de la structure.
*
* On utilise le résultat pour effectuer une analyse
* de flambage.
*
*******************************************************
*
TITRE 'CYLINDRE INFINI SOUS PRESSION EXTERNE';
OPTI DIME 2 ELEM SEG2 MODE FOUR 0;
GRAPH='N';
TEMPS ;

*
*----------- CONSTRUCTION DE LA GEOMETRIE -------------
*
PA=1000. 0.;PB=1000. 100.;PO1=0. 0.;PO2=0. 100.;
CYL=PA D 4 PB;
*
SI (NEG GRAPH 'N');
TRAC ECLA ELEM CYL;
FINSI;
*
MOD1=MODL CYL MECANIQUE ELASTIQUE COQ2;

*
*--- DECLARATION DE FOURIER NOHARM POUR LES OBJETS ----
*-------  QUI SERONT UTILISES POUR PLUSIEURS ----------
*--------------- NUMEROS D'HARMONIQUE -----------------
*
OPTI MODE FOUR NOHARM;
*
*
*-------- CONDITIONS AUX LIMITES SYMETRIQUES ----------
*
SYMB=SYMT CYL DEPL ROTA PA PO1  0.5;
SYMH=SYMT CYL DEPL ROTA PB PO2  0.5;
CDL=SYMB ET SYMH;
*
*----------- MATERIAU ET CARACTERISTIQUES -------------
*
PP = MANU CHPO CYL 1 P 1. ;
MAT =MATR MOD1 YOUN 20000. NU 0.;
CAR=CARB MOD1 EPAI 1.;
MAT=MAT ET CAR;
*
*----------- DECLARATION DE FOURIER MODE 0 ------------
*----------- POUR LE CALCUL DES CONTRAINTES -----------
*
OPTI MODE FOUR 0;

FP=PRES COQU MOD1 1. NORM ;
RIG=RIGI MOD1 MAT;
*
RIGADD = RIGI UT 1.E-4 PA ;
*
AAA = RIG ET CDL ET RIGADD ;
U=RESO AAA  FP ;
SIG=SIGMA U MOD1 MAT;
*
*----------- DECLARATION DE FOURIER MODE 2 ------------
*------------ POUR L'ANALYSE DE FLAMBAGE --------------
*
OPTI MODE FOUR 2;
*
PP = MANU CHPO CYL 1 P 1. ;
*
MKSI=KSIGMA MOD1 MAT (SIG * -1.)  FLAM;
RIG=RIGI MOD1 MAT ;
*
*------- RECHERCHE DE LA 1ERE FREQUENCE PROPRE --------
*
MODF=VIBR PROC (PROG 0.) (RIG ET CDL) MKSI;
LAMBDA1=((TIRE MODF FREQ RANG 1) * 6.28) ** 2;
*
MKP= KP PP MOD1 FLAM ;
MODF=VIBR PROC (PROG 0.) (RIG ET CDL) (MKSI ET MKP);
LAMBDA2=((TIRE MODF FREQ RANG 1) * 6.28) ** 2;
*
SAUT PAGE ;
SAUT 5 LIGN ;
MESS 'K(SIG) SEUL  : ON DOIT TROUVER LAMBDA=  6.66E-6';
MESS '               LE CALCUL DONNE LAMBDA=' LAMBDA1;
SAUT 2 LIGN ;
*
MESS 'K(SIG)+K(P)  : ON DOIT TROUVER LAMBDA=5E-6' ;
MESS '               LE CALCUL DONNE LAMBDA=' LAMBDA2;
SAUT 2 LIGN ;
*
TEMPS ;
*-------------  CODE DE FONCTIONNEMENT ----------------

ERR1=100*(ABS(6.66E-6 - LAMBDA1)/6.66E-6);
ERR2=100*(ABS(5E-6 - LAMBDA2)/5E-6);
ERRMAX=MAXI (PROG ERR1 ERR2);
SI (ERRMAX < 5 );
ERRE 0;
SINON ;
ERRE 5;
FINSI;

FIN;
```

Test four1 **Comments**

1. **Drawing of meshing with dispersed elements**

   TRAC ECLA ELEM CYL;

   The operator TRAC with keyword ECLA allows to represent the meshing with dispersed elements. Each element is represented with a homothetic rate (0.5 by default). The other keyword ELEM allows to indicate the local number of the elementary objects.

2. **Computation with FOURIER mode**

   OPTI MODE FOUR NOHARM;

   The computation mode is defined with the operator OPTI. The keyword NOHARM allows to create objects which coulb be used with any number of harmonic.

   OPTI MODE FOUR 0;

   All objects which are created after this command will be used only with harmonic 0. When the computation is done with harmonic 2, the previous objects are unavailable.

3. **Determination of natural frequencies**

   MODF = VIBR PROC (PROG 0.) (RIG ET CDL) MKSI;
   LAMBDA1 = ((TIRE MODF FREQ RANG 1) * 6.28) ** 2;

   The natural mode shapes are calculated which frequency value is close to zero. The type of the object MODF is called SOLUTION. MODF has different components among wich the frequency (component FREQ).
   In order to obtain the value of the frequency, we use the operator TIRE with keyword FREQ to indicate the type of the desired component and the keyword RANG to indicate the row of the moment. In this case there is only one shape, so the value of the row is one, but if there were two shapes, the first row would correspond to the lower frequency and the second to the higher frequency.

4. **Computation of pressure matrix**

   PP = MANU CHPO CYL 1 P 1. ;
   MKP = KP PP MOD1 FLAM ; The operator KP creates the pressure matrix joined to the work of

   pressure in large displacements. With keyword FLAM, the created object is a mass matrix.

Chapter 3

PLASTICITY

3.1 Test plas1 Description sheet

Test name plas1

Calculation type MECHANICS PLASTICITY 2D PLANE STRESS

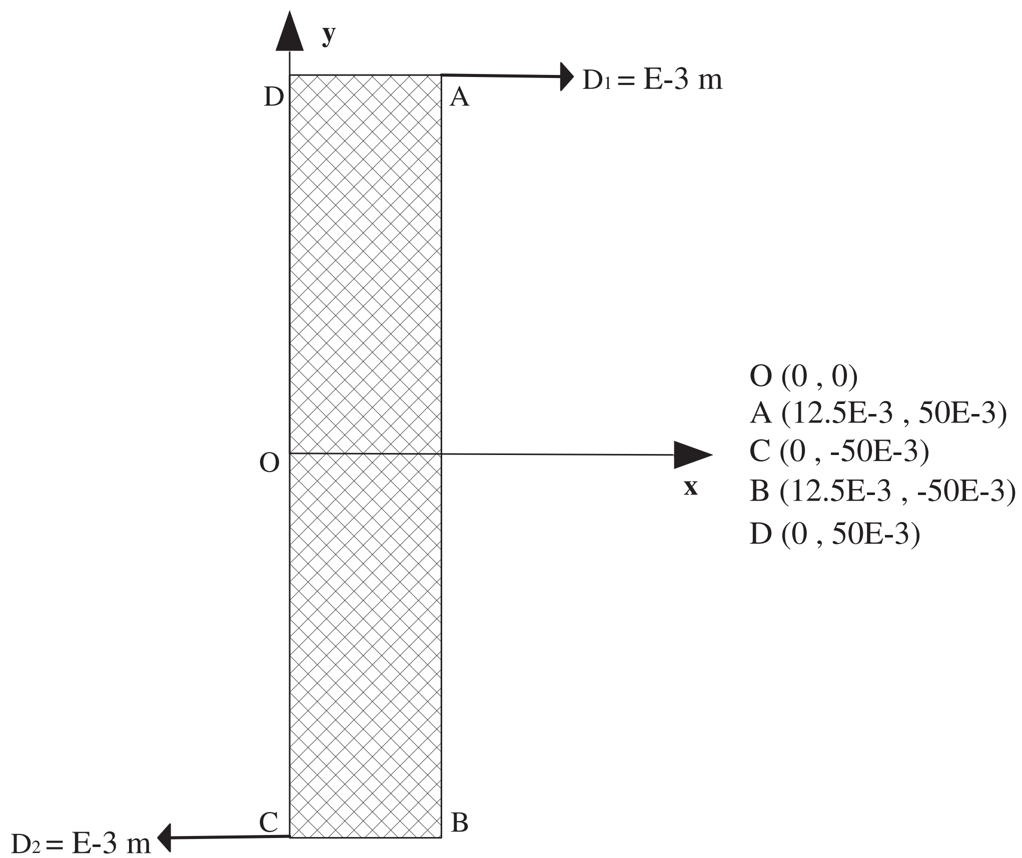
Finite element type COQ4

Topic
A rectangular-plate subjected to simple bending.

The structure is a rectangular-plate subjected to a simple bending. This bending is obtained by the fixed displacement (D) at the tips (see figure), wich is a function of time (T).
At the first step of time (0.02875), the strain range is elastic, at the second one (0.05), it is plastic. At both time steps, the elastic and the plastic theoretical bending moments are compared to the CASTEM computed values. These values are computed from the reactions. **Goal** Find the computed elastic and plastic bending

moment M1 and M2. The theoreticals values are :
M1 = 805 kN M2 = 1074 kN

Reference CASTEM Test NAFEMS : Analytical-Solution : MP1 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97' customer version **Model description**



E = $2.1\ 10^{11}\ Pa$
NU = 0.28
SYG = $483E6\ Pa$

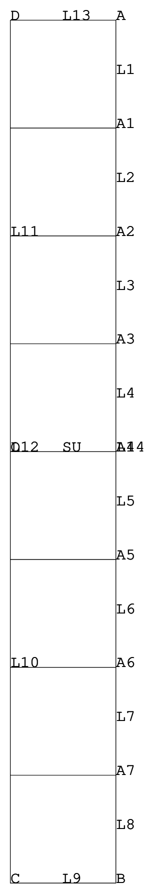
Test plas1

RESULTS

Results

	$M1$	$M2$
Reference	$805kN$	$1074KN$
Castem	$811.08kN$	$1085.2kN$

CASTEM FIGURES



A RECTANGULAR-PLATE SUBJECTED TO SIMPLE FLEXION

```
*          Test Plas1.dgibi: Jeux de données
*          ---------------------------------
*
* CAS TEST DU 91/10/24     PROVENANCE : MILL
* CAS TEST DU 91/10/15     PROVENANCE : STRU
OPTION ECHO 0;
GRAPH='N';
SAUT PAGE ;
*
*******************************************************
*  TEST PLAS1
*
* Sortie du domaine élastique et phase plastique
* (comportement élasto-plastique parfait)
*
* Une plaque rectangulaire est soumise à une flexion
* pure. La flexion est imposée par le déplacement des
* extrémités de la plaque (voir schéma ci-dessous) en
* fonction du temps.
*
* Au premier pas de temps (0.02875) on se trouve dans
* le domaine élastique et au deuxième pas de temps
* (0.05) on se trouve dans le domaine plastique. Au
* deux pas de temps, les moments fléchissants calculés
* à partir des forces de réaction sont comparés avec
* les résultats théoriques.
*
*******************************************************
*
*     D .____. A    ----> D1
*       |    |
*       |    |
*       .____. A1
*       |    |
*       |    |
*       .____. A2
*       |    |
*       |    |
*       .____. A3
*       |    |
*       |    |
*     O .____. A4
*       |    |
*       |    |
*       .____. A5
*       |    |
*       |    |
*       .____. A6
*       |    |
*       |    |
*       .____. A7
*       |    |
*       |    |
*     C .____. B    <---- D2
*
*------------------------------------------------------
* LES POINTS A1 A A7 SERVIRONT A FAIRE VARIER LE
* DEPLACEMENT IMPOSE LINEAIREMANT DE A (D1) A B (D2)
*------------------------------------------------------
*
*******************************************************
TITRE 'PLAQUE RECTANGULAIRE EN FLEXION PURE' ;
OPTION  ECHO 1 ;
OPTION DIME 2 ELEM QUA4 MODE PLAN CONT ;
TEMPS ;
*
*-------------- D{finition de la g{om{trie ------------
*
* O = 0 0 ;
* A = 12.5E-3 50E-3 ;
* B = 12.5E-3 -50E-3 ;
C = 0 -50E-3 ;
D = 0 50E-3 ;
*
A1 = 12.5E-3 37.5E-3 ;
A2 = 12.5E-3 25E-3 ;
A3 = 12.5E-3 12.5E-3 ;
A4 = 12.5E-3 0 ;
A5 = 12.5E-3 -12.5E-3 ;
A6 = 12.5E-3 -25E-3 ;
A7 = 12.5E-3 -37.5E-3 ;
*
L1 = A DROIT 1 A1 ;
L2 = A1 DROIT 1 A2 ;
L3 = A2 DROIT 1 A3 ;
L4 = A3 DROIT 1 A4 ;
L5 = A4 DROIT 1 A5 ;
L6 = A5 DROIT 1 A6 ;
L7 = A6 DROIT 1 A7 ;
L8 = A7 DROIT 1 B ;
*
L14 = L1 ET L2 ET L3 ET L4 ET L5 ET L6 ET L7 ET L8 ;
*
L9 = B DROIT 1 C ;
L10 = C DROIT 4 O ;
L11 = O DROIT 4 D ;
L12 = L10 ET L11 ;
L13 = D DROIT 1 A ;
*
SU = L14 L9 L12 L13 DALL PLAN ;
ELIM 0.01 SU ;
SI(NEG GRAPH 'N');
   TRACE SU ;
FINSI;
*
*------- Définition des conditions aux limites --------
*----------- et des déplacements imposés -------------
*
CL1 = BLOQ L12 UX  ;
CL1B = BLOQ O UY ;
CL2 = BLOQ A UX ;
CL3 = BLOQ B UX ;
CL = CL1 ET CL1B ET CL2 ET CL3 ;
*
D1 = DEPI CL2 1E-3 ;
D2 = DEPI CL3 -1E-3 ;
DEP = D1 ET D2 ;
*
*-------------------------------------
* Exemple : EN A1 , D = 7/8 D1 + 1/8 D2
*-------------------------------------
*
*--------- Les déplacements en chaque noeud -----------
*----------- est une relation linéaire des ------------
*------------ d{éplacements des extrémités ------------
*------- Création de la rigidité correspondante -------
*
R1 = RELA 8 UX A1 - 7 UX A - 1 UX B ;
R2 = RELA 8 UX A2 - 6 UX A - 2 UX B ;
R3 = RELA 8 UX A3 - 5 UX A - 3 UX B ;
R4 = RELA 8 UX A4 - 4 UX A - 4 UX B ;
R5 = RELA 8 UX A5 - 3 UX A - 5 UX B ;
R6 = RELA 8 UX A6 - 2 UX A - 6 UX B ;
R7 = RELA 8 UX A7 - 1 UX A - 7 UX B ;
R = R1 ET R2 ET R3 ET R4 ET R5 ET R6 ET R7 ;
*
```

```
*-------- Utilisation de l'opérateur NONLIN -----------
*-- Les objets utilisés par la procédure NONLIN sont --
*------- définis dans les lignes qui suivent ----------
*
MO= MODL SU MECANIQUE ELASTIQUE PLASTIQUE PARFAIT QUA4;

*
*----- Définition du matériau et de la rigidité -------
*----------- SIGY est la limite élastique -------------
*
MAT1 = MATR MO YOUN 2.1E11 SIGY 483E6 NU 0.3 ;
RI1 = RIGI MO MAT1 ;
RIG1 = RI1 ET CL ET R ;
*
*------- Création d'une relation linéaire entre -------
*------------- le déplacement et le temps -------------
*
LI1 = PROG 0. 1. ;
LI2 = PROG 0. 1. ;
EV = EVOL MANU T LI1 F(T) LI2 ;
*
* Remarque : au temps T, le d{placement vaut F(T)*D .
*----------------------------------------------------
*
CHA1 = CHAR 'DIMP' DEP EV ;
*
*------- Création de la liste des pas de calcul -------
*----- 0.02875 est le dernier incrément élastique -----
*- 0.05 est un incrément plastique (calcul analytique)
*
LIS1 = PROG 0.02875 0.05 ;
*
*----------- Calcul par l'op{rateur NONLIN ------------
*
TAB1 = TABLE ;
TAB1.'BLOCAGES_MECANIQUES' = CL 'ET' R;
TAB1.'MODELE' = MO;
TAB1.'CHARGEMENT' = CHA1;
TAB1.'CARACTERISTIQUES' = MAT1;
TAB1.'TEMPS_CALCULES' = LIS1;

PASAPAS TAB1 ;
*
*--------- calcul et impression des résultats ---------
*
REA1 = REAC RIG1 (TAB1.DEPLACEMENTS.1) ;
REA2 = REAC RIG1 (TAB1.DEPLACEMENTS.2) ;
*
R11 = EXTR REA1 FX A ;
R21 = EXTR REA2 FX A ;
R12 = EXTR REA1 FX A1 ;
R22 = EXTR REA2 FX A1 ;
R13 = EXTR REA1 FX A2 ;
R23 = EXTR REA2 FX A2 ;
R14 = EXTR REA1 FX A3 ;
R24 = EXTR REA2 FX A3 ;
*
* Pour obtenir les moments fléchissants, les réactions
* - obtenues sont multipliées par leur bras de levier -
*
M1 = (2.E-3)*((0.050*R11)+(0.0375*R12)+(0.025*R13)
     +(0.0125*R14)) ;
M2 = (2.E-3)*((0.050*R21)+(0.0375*R22)+(0.025*R23)
     +(0.0125*R24)) ;
*
SAUT PAGE ; SAUT 2 LIGN ;
*
MESS '                    MOMENT THEORIQUE
MOMENT CALCULE';
SAUT 1 LIGN ;
MESS 'INCREMENT ELASTIQUE    M1= 805 KN
M1= 'M1 'KN';
SAUT 1 LIGN ;
MESS 'INCREMENT PLASTIQUE    M2= 1074 KN
M2= 'M2 'KN';
*
TEMPS ;
*         CODE FONCTIONNEMENT
MR1=805.  ;
MR2=1074. ;
RESI1=ABS((M1-MR1)/MR1);
RESI2=ABS((M2-MR2)/MR2);
SI((RESI1 <EG 5E-2) ET (RESI2 <EG 5E-2));
  ERRE  0;
SINO;
  ERRE  5;
FINSI;
FIN  ;
```

Test plas1 **Comments**

1. BOUNDARY CONDITIONS

```
CL1 = BLOQ L12 UX  ;  CL1B = BLOQ O UY ;  CL2 = BLOQ A UX    ;  CL3 = BLOQ B UX ;
CL = CL1 ET CL1B ET CL2 ET CL3 ;
```

The BLOQUE operator constructs the RIGIDITE stiffness, which is associated to conditions of imposed values on the unknowns of a discretized problem. Later, this stiffness will have to be added to that of the structure.
The non-zero values to be imposed must also be provided in a CHPOINT type object built by means of the DEPIMP operator. For example here :

```
D1 = DEPI CL2 1E-3 ;   D2 = DEPI CL3 -1E-3 ;   DEP = D1 ET D2 ;
```

Thus, The DEPI operator specifies the value of certain relationships. CL2 and CL3 are the RIGIDITE, type object, of BLOCAGE subtype, defining the conditions imposed on the degrees of freedom . Here 1E-3 and -1E-3 , are the values to be imposed on certain displacement relations.

```
R1 = RELA 8 UX A1 - 7 UX A - 1 UX B ;   R2 = RELA 8 UX A2 - 6 UX A - 2 UX B ;
R3 = RELA 8 UX A3 - 5 UX A - 3 UX B ;   R4 = RELA 8 UX A4 - 4 UX A - 4 UX B ;
R5 = RELA 8 UX A5 - 3 UX A - 5 UX B ;   R6 = RELA 8 UX A6 - 2 UX A - 6 UX B ;
R7 = RELA 8 UX A7 - 1 UX A - 7 UX B ;   R = R1 ET R2 ET R3 ET R4 ET R5 ET R6 ET R7 ;
```

The RELATION operator enables the user to construct the stiffness associated with a linear relation between the unknowns named Ui, weighted by (possible) coefficients, for respective mesh nodes (here it is UX A1, UX B , UX A2, UX B,.......). The DEPIMP operator allows the user to impose a non-zero value on the relation. For example here at a point A1 the displacement D is :
$D = 7/8D1 + 1/8D2$

2. CALCULATION WITH THE NONLIN OPERATOR
PROCEDURE PASAPAS

```
TAB1 = TABLE ;                    TAB1.'BLOCAGES_MECANIQUES' = CL 'ET' R;
TAB1.'MODELE' = MO;               TAB1.'CHARGEMENT' = CHA1;
TAB1.'CARACTERISTIQUES' = MAT1;   TAB1.'TEMPS_CALCULES' = LIS1;
PASAPAS TAB1 ;
```

The (NONLIN) PASAPAS procedure enables the user to carry out an incremental non linear calculation. The non linearity can result either from the material (plasticity) or from the large displacements, or else from both. Here the non linearity resut from the material.
As input, TAB1 is used to define the options and the calculation parameters. The indices of the TAB1 object are words (written in words), their list is precedently specified.

3. RESULTS AND OUTPUT

```
REA1 = REAC RIG1 (TAB1.DEPLACEMENTS.1) ;  REA2 = REAC RIG1 (TAB1.DEPLACEMENTS.2) ;
```

The reaction operator REAC constructs the reaction force from the solution field TAB1.DEPLACEMENT.1 and TAB1.DEPLACEMENTS.2, of CHPOINT types, resulting from a linear system on the first member of which conditions have been imposed. allowing the verification of these conditions.
Thus, for to obtain the bending moments, these reactions are multiplyed by their lever arm :

```
M1 = (2.E-3)*((0.050*R11)+(0.0375*R12)+(0.025*R13)+(0.0125*R14)) ;
M2 = (2.E-3)*((0.050*R21)+(0.0375*R22)+(0.025*R23)+(0.0125*R24)) ;
```

3.2 Test plas2 Description sheet

Test name plas2

Calculation type MECHANIC PLASTICITY 2D AXI

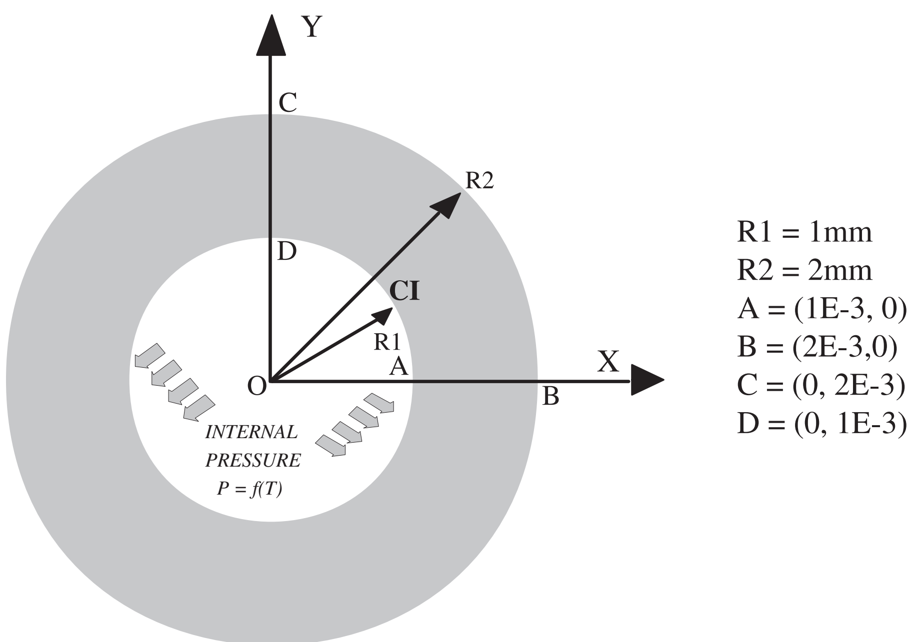
Finite element type COQ4

Topic
A Spheric-Solid, subjected to an internal pressure.

The structure is a Hollow-Spheric-Solid subjected to an internal pressure. The geometry is axisymetrical, the inner radius is R1 = 1mm and the outer radius is R2 = 2mm.
The internal pressure P varies as a function of time : at 0s $P = 100MPa$ and at 1000s $P = 358.9MPa$
This non linear evolution (resulting from the material) responds in a plastic manner up to
R = 1.5 mm. time (T).

Goal Find the computed radial displacements $U_r(A)_{t0}$ and $U_r(A)_{t1000}$ The theoretical values are :
$U_r(A)_{t0} = 0.4m$ $\quad U_r(A)_{t1000} = 2.83m$

Reference CASTEM Test NAFEMS : Analytical-Solution : MP2 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97' customer version **Model description**



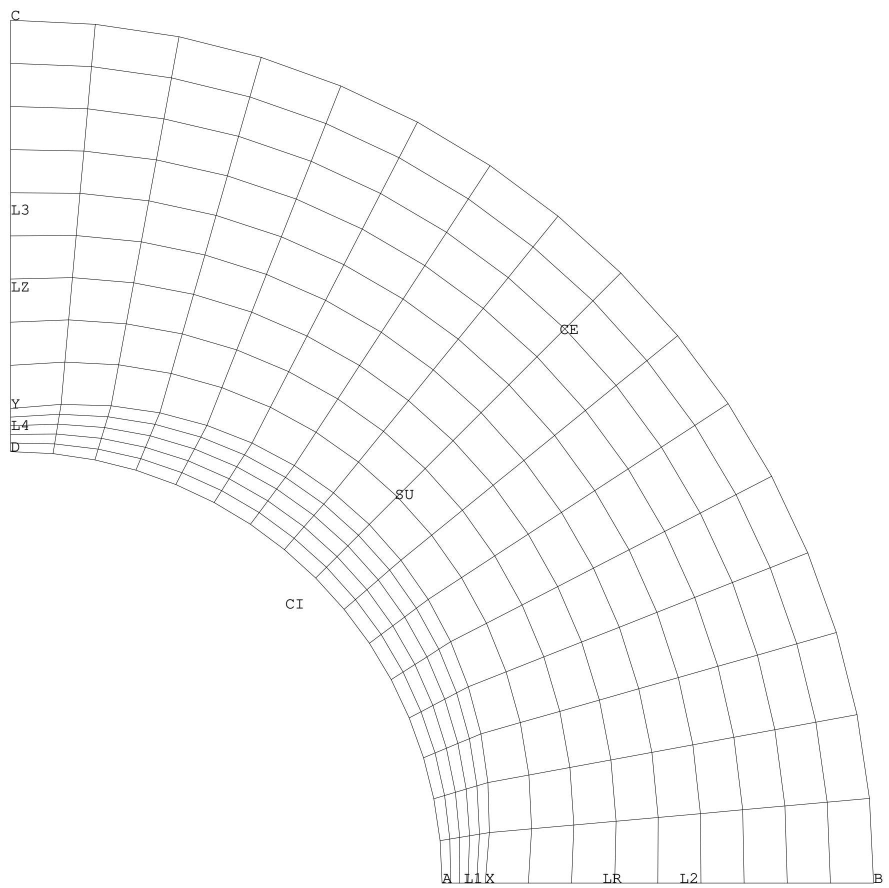
E = $2.1\ 10^{11}\ Pa$
NU = 0.3
SYG = $300E6\ Pa$

Test plas2 **Results**

RESULTS

	$U_r(A)_{t0}$	$U_r(A)_{t1000}$
Reference	0.4 m	2.83 m
Castem	0.39821 m	1085.2 m

CASTEM FIGURES



A SOLID-SPHERIC SUBJECTED TO INTERNAL PRESSURE

```
*          Test Plas2.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 1;
GRAPH='N';
SAUT PAGE ;
*******************************************************
* TEST PLAS2
*
* Sortie du domaine élastique et phase plastique
* (comportement élasto-plastique parfait)
* Calcul axisymétrique avec prise en compte des
* symétries.
*
* Une sphère de rayon extérieur égal à 2 mm et de rayon
* intérieur égal à 1 mm (centre O et contour ABCD) est
* soumise à une pression interne.
*
* La pression évolue en fonction du temps de 100 MPa à
* 0 s jusqu'à 358.9 MPa à 1000 s.
*
* Cette évolution de la pression entraine un
* comportement plastique jusqu'en R=1.5 mm.
*
* Les déplacements radiaux à 0 s et à 1000 s
* sont comparés aux valeurs théoriques (0.4 microns
* et 2.83 microns)
*
*******************************************************
TITRE 'PLAS2 - SPHERE SOUS PRESSION INTERNE';
OPTION DIME 2 ELEM QUA4 MODE AXIS ;
GRAPH='N';
TEMPS ;
*
*------------- Définition de la géométrie -------------
*
O = 0 0 ;
A = 1E-3 0 ;
B = 2E-3 0 ;
C = 0 2E-3 ;
D = 0 1E-3 ;
*
*-----------------------------------------------------
* La peau interne est maillée de façon plus fine pour
* un meilleur calcul des contraintes radiales, les
* points X et Y sont introduits à cet effet.
*-----------------------------------------------------
*
X = 1.1E-3 0 ;
Y = 0 1.1E-3 ;
*
L1 = A DROIT 5 X ;
L2 = X DROIT 9 B ;
LR = L1 ET L2 ;
CE = CERC 16 B O C ;
L3 = C DROIT 9 Y ;
L4 = Y DROIT 5 D ;
LZ = L3 ET L4 ;
CI = CERC 16 D O A ;
*
SU = LR CE LZ CI DALL PLAN ;
*
SI (NEG GRAPH 'N');
TRACE SU ;
FINSI;
*
*------ Définition des conditions aux limites  --------
*----------- (conditions de symétrie) -----------------
```

```
*
CL1 = BLOQ LR UZ ;
CL2 = BLOQ LZ UR ;
CL = CL1 ET CL2 ;
*
*--------- Utilisation de la procedure PASAPAS --------
*-- Les objets utilisés par la procédure PASAPAS sont -
*--------- définis dans les lignes qui suivent --------
*
MO= MODL SU MECANIQUE ELASTIQUE PLASTIQUE PARFAIT QUA4;
*
*----------------------------
* SIGY est la limite élastique
*----------------------------
*
*------------- Définition du matériau -----------------
*
MAT1 = MATR MO YOUN 2E11 SIGY 300E6 NU 0.3;
*
*------ Définition en Pa de la pression unitaire ------
*
FO = PRES MASS MO 1E6 CI ;
*
*---------- au pas de temps T, la pression vaut F(T)*FO
*
LI1 = PROG 0. 1000. ;
LI2 = PROG 0. 1000. ;
EV = EVOL MANU T LI1 F(T) LI2 ;
CHA1 = CHAR MECA FO EV ;
*
*-- Liste des pas de calcul, 100. MPa est la pression -
*--- de début de plastification, 358.9 MPa engendre  --
*----- un comportement plastique jusqu'en R=1.5 mm ----
*
LIS1 = PROG 100. 358.9 ;
TAB1 = TABLE ;
TAB1.'BLOCAGES_MECANIQUES' = CL;
TAB1.'MODELE' = MO;
TAB1.'CARACTERISTIQUES' = MAT1;
TAB1.'CHARGEMENT' = CHA1;
TAB1.'TEMPS_CALCULES' = LIS1;

PASAPAS TAB1 ;
*
*-------- Calcul et impression des résultats ----------
*
RE1 = (TAB1.DEPLACEMENTS.1) ;
RE2 = (TAB1.DEPLACEMENTS.2) ;
*
V1 = EXTR RE1 UR A ;
V1 = 1E6*V1 ;
V2 = EXTR RE2 UR A ;
V2 = 1E6*V2 ;
*
SAUT PAGE ; SAUT 2 LIGN ;
*
MESS '           EN R=1MM  :  DEPLACEMENT THEORIQUE
 CALCULE' ;
SAUT 2 LIGN ;
MESS 'INCREMENT ELASTIQUE       UR= 0.40 MICRONS
UR= 'V1 'MICRON';
SAUT 1 LIGN ;
MESS 'INCREMENT PLASTIQUE       UR= 2.83 MICRONS
UR= 'V2 'MICRON';
TEMPS ;
*------------ Code de fonctionnement -----------------

ERR1=100*(ABS(0.4 - V1)/0.4);
```

```
ERR2=100*(ABS(2.83 - V2)/2.83);
LIST1=PROG ERR1 ERR2;
ERRMAX=MAXI (PROG ERR1 ERR2);
SI (ERRMAX < 5);
ERRE 0;
SINON;
ERRE 5;
FINSI;

FIN;
```

Test plas2 **Comments**

LOAD DEFINITION RESULTING FROM AN INTERNAL PRESSURE

```
FO = PRES MASS MO 1E6 CI ;
```

The PRES operator calculates the forces which are equivalent to a pressure applied on an object. Here this operator is connected to key word MASS, referring to the type of solid element on which the pressure is applied.
MO is the object on which the pressure is applied (MMODEL type) and 1E6 is a pressure algebraic value (FLOTTANT type). CI is the mesh on which the pressure is applied (MAILLAGE type).
For solid elements, such as those used in this problem, the pressure is assumed to be directed towards the inside of the solid. In case of a depression, it is required that a field of a negative pressure points have to be supplied.

```
LI1 = PROG 0. 1000. ;
LI2 = PROG 0. 1000. ;
EV = EVOL MANU T LI1 F(T) LI2 ;
CHA1 = CHAR MECA FO EV ;
```

EV is time description of the loading (EVOLUTION type) function containing in abscissa the times (in chronological order) and in ordinate the values F(T) of the function F for each value of the times T.
Thus, The CHAR operator constructs a CHAR1 object of CHARGEMENT type, of FORCE subtype which contains the space and time description of the equivalent loading.

3.3 Test plas4 Description sheet

Test name plas4

Calculation type MECHANICS PLASTICITY 3D

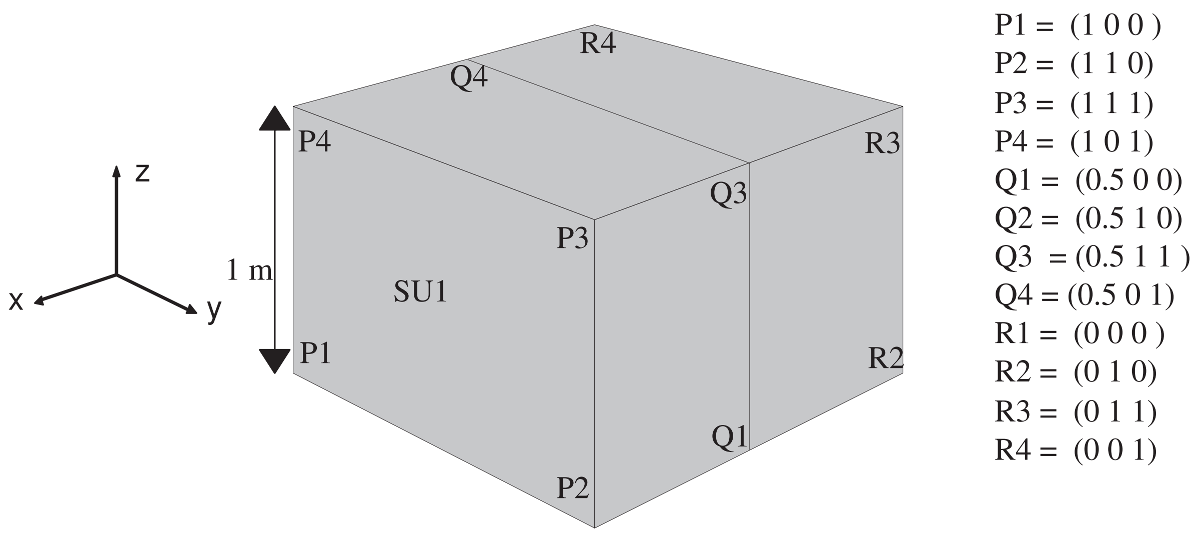
Finite element type CUB8

Topic
A Cubical solid, subjected to tensile test.

The structure is a cubical-Solid composed by two element. It is subjected to a tensile test.
For this test, The Y and Z displacement values are fixed at zero value and the X displacement value is imposed on the tips : SU1 and SU2 surfaces.

Goal Find the computed stress Smxx, Smyy and Smzz at 4 time steps (T1, T2, T3 and T4) on the first Gaussian point of the first cubical element. These CASTEM computed values are compared to the theorical values.

Reference CASTEM Test NAFEMS : Analytical-Solution : MP4 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97' customer version **Model description**



Cubic Solid subjected to Tensile Test Samples
Ux = f(t) , Uy = Uz = 0

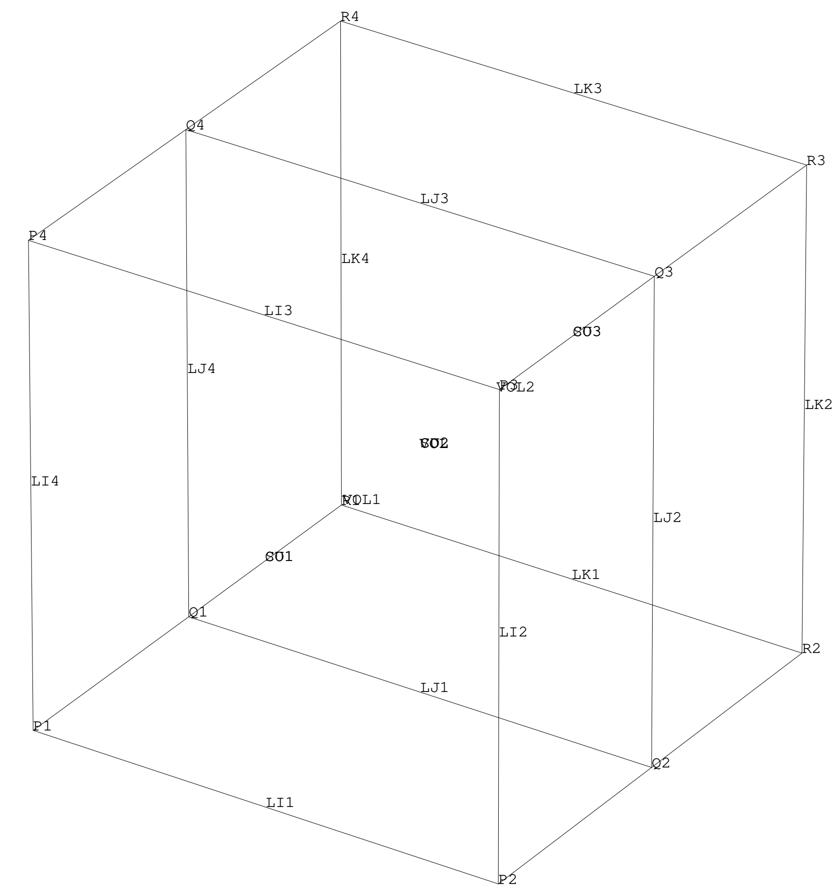
E = $2.5\ 10^{11}\ Pa$
NU = 0.25
SYG = $5E6\ Pa$

Test plas4 **Results**

RESULTS

Evolution Parameter	$T1$	$T2$	$T3$	$T4$
Reference	$Txx = 7.5E6$	$Txx = 11.66E6$	$Txx = 4.16E6$	$Txx = -3.33E6$
	$Tyy = 2.5E6$	$Tyy = 6.66E6$	$Tyy = 4.16E6$	$Tyy = 1.66E6$
	$Tzz = 2.5E6$	$Tzz = 6.66E6$	$Tzz = 4.16E6$	$Tzz = 1.66E6$
Castem	$Txx = 7.5E6$	$Txx = 11.66E6$	$Txx = 4.16E6$	$Txx = -3.33E6$
	$Tyy = 2.5E6$	$Tyy = 6.66E6$	$Tyy = 4.16E6$	$Tyy = 1.6E6$
	$Tzz = 2.5E6$	$Tzz = 6.66E6$	$Tzz = 4.16E6$	$Tzz = 1.6E6$

CASTEM FIGURES



CUBE EN TRACTION UNIAXIALE

```
*          Test Plas4.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
GRAPH='N';
SAUT PAGE;
*****************************************************
*     CUBE EN TRACTION UNIAXIALE
*
*     TEST ELEMENTAIRE DU GROUPE DE TRAVAIL
*     'STATIQUE NON LINEAIRE' COMMISSION VPCS
*
*     LE CALCUL MARCHE ET DONNE EXACTEMENT LES
*     RESULTATS THEORIQUES LE 05/05/90
*
*     Un cube constitué de 2 éléments CUB8 est soumis à
*     une traction uniaxiale (blocages des déplacements
*     suivant Y et Z et déplacement imposé suivant X
*     aux deux extrémités (surfaces SU1 et SU2))
*
*     Les contraintes obtenues sont comparées aux
*     résultats théoriques et donnent les résultats
*     précisés plus bas.
*
*****************************************************
*

SAUT PAGE;
TITRE 'CUBE EN TRACTION UNIAXIALE';
OPTION DIME 3 ECHO 0;
TEMPS ;
DENS 1;
*----------------------------------------------------
***** DEFINITION DE LA GEOMETRIE
*----------------------------------------------------
A = 1.;
A1 = -1.;
A2 = .5;
B = 1.;
C = 1.;
P1 = A 0 0; Q1 = A2 0 0; R1 = 0 0 0;
P2 = A B 0; Q2 = A2 B 0; R2 = 0 B 0;
P3 = A B C; Q3 = A2 B C; R3 = 0 B C;
P4 = A 0 C; Q4 = A2 0 C; R4 = 0 0 C;
TR = A1 0 0;
OPTION ELEM SEG2;
LI1 = P1 DROIT 1 P2;
LI2 = P2 DROIT 1 P3;
LI3 = P3 DROIT 1 P4;
LI4 = P4 DROIT 1 P1;
CO1 = LI1 ET LI2 ET LI3 ET LI4;
LJ1 = Q1 DROIT 1 Q2;
LJ2 = Q2 DROIT 1 Q3;
LJ3 = Q3 DROIT 1 Q4;
LJ4 = Q4 DROIT 1 Q1;
CO2 = LJ1 ET LJ2 ET LJ3 ET LJ4;
LK1 = R1 DROIT 1 R2;
LK2 = R2 DROIT 1 R3;
LK3 = R3 DROIT 1 R4;
LK4 = R4 DROIT 1 R1;
CO3 = LK1 ET LK2 ET LK3 ET LK4;
OPTION ELEM QUA4;
SU1 = SURFACE CO1 PLANE;
SU2 = SURFACE CO2 PLANE;
SU3 = SURFACE CO3 PLANE;
OPTION ELEM CUB8;
VOL1 = SU1 VOLU 1 SU2;
VOL2 = SU2 VOLU 1 SU3;
VOL = VOL1 ET VOL2;
```

```
ELIM .001 VOL;
*----------------------------------------------------
***** TRACE
*----------------------------------------------------
OEIL = (30 20 20);
SI(NEG GRAPH 'N');
   TRAC OEIL VOL;
FINSI;
*----------------------------------------------------
***** CONDITIONS AUX LIMITES
*----------------------------------------------------
CL1 = BLOQUE SU3 UX;
CL2 = BLOQUE VOL UY UZ;
CL3 = BLOQUE SU2 UX;
CL4 = BLOQ SU1 UX;
CL = CL1 ET CL2 ET CL3 ET CL4;
*----------------------------------------------------
* VALEURS IMPOSEES DES DEPLACEMENTS
*----------------------------------------------------
EPS0 = .25E-4;
COEF1 = A * EPS0 * .5;
COEF2 = A * EPS0;
D1 = DEPI CL3 COEF1;
D2 = DEPI CL4 COEF2;
DEP = D1 ET D2;
*----------------------------------------------------
***** CALCUL MODELE CARACTERISTIQUES
*----------------------------------------------------
MO=MODL VOL MECANIQUE ELASTIQUE PLASTIQUE PARFAIT CUB8;
MAT1 = MATR MO YOUN 2.5E11 SIGY 5.0E6 NU .25;
*----------------------------------------------------
***** CHAMPS RELATIFS A L'OPERATEUR NONLIN
*----------------------------------------------------
LP1 = PROG 0. 1. 2. 3. 4.;
LP2 = PROG 0. 1. 2. 1. 0.;
EV = EVOL MANU T LP1 F(T) LP2;
CHA1 = CHAR DIMP DEP EV;
LIS1 = PROG 1. 2. 3. 4.;
*----------------------------------------------------
*****  CALCUL NON LINEAIRE
*----------------------------------------------------
TAB1 = TABLE;
TAB1.'CARACTERISTIQUES' = MAT1;
TAB1.'CHARGEMENT' = CHA1;
TAB1.'MODELE' = MO;
TAB1.'TEMPS_CALCULES' = LIS1;
TAB1.'BLOCAGES_MECANIQUES' = CL;
PASAPAS TAB1;
*----------------------------------------------------
***** EXTRACTION DES RESULTATS
*----------------------------------------------------
SI1 = TAB1.CONTRAINTES.1;
SI2 = TAB1.CONTRAINTES.2;
SI3 = TAB1.CONTRAINTES.3;
SI4 = TAB1.CONTRAINTES.4;
SXX1 = EXTR SI1 SMXX 1 1 1;
SYY1 = EXTR SI1 SMYY 1 1 1;
SZZ1 = EXTR SI1 SMZZ 1 1 1;
SXX2 = EXTR SI2 SMXX 1 1 1;
SYY2 = EXTR SI2 SMYY 1 1 1;
SZZ2 = EXTR SI2 SMZZ 1 1 1;
SXX3 = EXTR SI3 SMXX 1 1 1;
SYY3 = EXTR SI3 SMYY 1 1 1;
SZZ3 = EXTR SI3 SMZZ 1 1 1;
SXX4 = EXTR SI4 SMXX 1 1 1;
SYY4 = EXTR SI4 SMYY 1 1 1;
SZZ4 = EXTR SI4 SMZZ 1 1 1;
*----------------------------------------------------
```

```
***** AFFICHAGE DES RESULTATS
*-------------------------------------------------------
SAUT PAGE;
*////////// PARAMETRE D'EVOLUTION  T : 1 //////////';
*-------------------------------------------------------
MESS '                T = 1 ';
MESS ' RESULTATS THEORIQUES : SIGMAXX =  7.5 E 6' ;
MESS '                        SIGMAYY =  2.5 E 6' ;
MESS '                        SIGMAZZ =  2.5 E 6' ;
MESS '                                            ' ;
MESS ' RESULTATS CALCULES   : SIGMAXX = 'SXX1 ;
MESS '                        SIGMAYY = 'SYY1 ;
MESS '                        SIGMAZZ = 'SZZ1 ;
SAUT 1 LIGNE ;
*
*////////// PARAMETRE D'EVOLUTION  T : 2 //////////';
*-------------------------------------------------------
MESS '                T = 2 ';
MESS ' RESULTATS THEORIQUES : SIGMAXX = 11.66 E 6' ;
MESS '                        SIGMAYY =  6.66 E 6' ;
MESS '                        SIGMAZZ =  6.66 E 6' ;
MESS '                                             ' ;
MESS ' RESULTATS CALCULES   : SIGMAXX = 'SXX2 ;
MESS '                        SIGMAYY = 'SYY2 ;
MESS '                        SIGMAZZ = 'SZZ2 ;
SAUT 1 LIGNE ;
*
*////////// PARAMETRE D'EVOLUTION  T : 3 //////////';
*-------------------------------------------------------
MESS '                T = 3 ' ;
MESS ' RESULTATS THEORIQUES : SIGMAXX =  4.16 E 6' ;
MESS '                        SIGMAYY =  4.16 E 6' ;
MESS '                        SIGMAZZ =  4.16 E 6' ;
MESS '                                             ' ;
MESS ' RESULTATS CALCULES   : SIGMAXX = 'SXX3 ;
MESS '                        SIGMAYY = 'SYY3 ;
MESS '                        SIGMAZZ = 'SZZ3 ;
SAUT 1 LIGNE ;
*
*////////// PARAMETRE D'EVOLUTION  T : 4 //////////';
*-------------------------------------------------------
MESS '                T = 4 ' ;
MESS ' RESULTATS THEORIQUES : SIGMAXX = -3.33 E 6' ;
MESS '                        SIGMAYY =  1.66 E 6' ;
MESS '                        SIGMAZZ =  1.66 E 6' ;
MESS '                                             ' ;
MESS ' RESULTATS CALCULES   : SIGMAXX = 'SXX4 ;
MESS '                        SIGMAYY = 'SYY4 ;
MESS '                        SIGMAZZ = 'SZZ4 ;
MESS '                                    ' ;
TEMPS ;
*                CODE DE FONCTIONNEMENT
SREF=1.66E6;RESI=ABS((SYY4-SREF)/SREF);
SI(RESI <EG 5E-2);
   ERRE  0;
SINO;
   ERRE  5;
FINSI;
FIN;
```

3.4 Test plas5 Description sheet

Test name plas5

Calculation type MECHANICS PLASTICITY 2D PLANE STRESSE

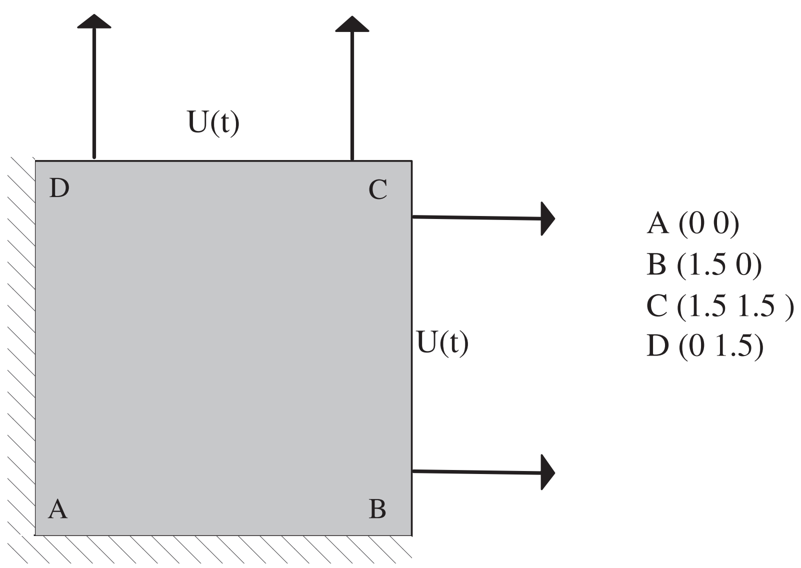
Finite element type QUA4

Topic
A solid plate, subjected to imposed displacement.

The structure is a solid plate, it is subjected to imposed displacement from two sides and blocked from the two others (see figure).
For this test, the goal is to study the DRUCKER-PRAGER elastic-plastic behaviour.

Goal Find the computed stress Smxx,and Smyy on the first Gaussian point of the first finite element. These CASTEM computed values are compared to the theoricals values.

Reference CASTEM Test NAFEMS : Analytical-Solution : MP4 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97' customer version **Model description**



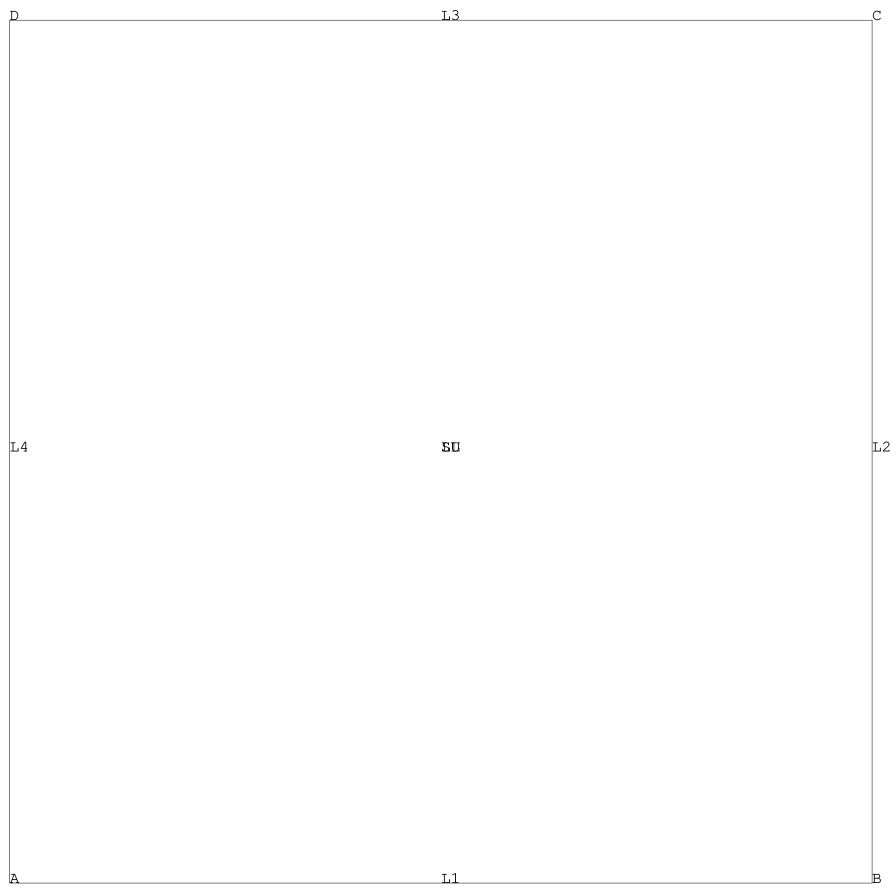
E	=	$2.5\ 10^{11}\ Pa$
NU	=	0.25
SYG	=	$5E6\ Pa$

Test plas5 **Results**

RESULTS

DRUCKER-PRAGER STRESSES	$Smxx$	$Smyy$
Reference	$1.8717Pa$	$1.8717Pa$
Castem	$1.8717Pa$	$1.8717Pa$

CASTEM FIGURES



A PLATE WITH CROSS SECTIONAL SUBJECTED TO BI-TENSILE TEST

```
*           Test Plas5.dgibi: Jeux de données        *
*           ---------------------------------        *
*                                                    *
OPTION ECHO 0;
GRAPH='N';
SAUT PAGE ;
*
******************************************************
*  TEST PLAS5
*
* Sortie du domaine élastique et phase plastique
* (comportement élasto-plastique modèle DRUCKER-PRAGER)
*
* Une plaque carrée est soumise à un déplacement imposé
* sur deux de ses bords et fixée sur les deux autres.
* On compare les contraintes à la solution exacte.
*
******************************************************
*
*        ^    ^
*        |    |  d(t)
*        |    |
*
*     D .____. C    ----> d(t)
*        |    |
*        |    |
*     A .____. B    ----> d(t)
*
*
******************************************************

TITRE 'PLAQUE CARRE EN BI-TRACTION PURE ';
OPTION ECHO 0 ;
OPTION DIME 2 ELEM QUA4 MODE PLAN CONT ;
TEMPS ;
*-----------------------------------------------------
*           DEFINITION DE LA GEOMETRIE
*-----------------------------------------------------
A = 0. 0. ;
B = 1.5 0. ;
C = 1.5 1.5 ;
D = 0. 1.5 ;
*
L1 = A DROIT 1 B  ;
L2 = B  DROIT 1 C  ;
L3 = C  DROIT 1 D  ;
L4 = D  DROIT 1 A  ;
LL = L1 ET L2 ET L3 ET L4 ;
SU = SURF LL PLAN ;
SI(NEG GRAPH 'N');
   TRACE  QUAL SU ;
FINSI;
*-----------------------------------------------------
*         DEFINITION DES CONDITIONS AUX LIMITES
*              ET DES DEPLACEMENTS IMPOSES
*-----------------------------------------------------
CL4 = BLOQ L4 UX  ;
CL2 = BLOQ L2 UX  ;
CL3 = BLOQ L3 UY  ;
CL1 = BLOQ L1 UY  ;
CL = CL1 ET CL2 ET CL3 ET CL4 ;
*
D3 = DEPI CL3 1.5E-4 ;
D2 = DEPI CL2 1.5E-4 ;
DEP = D2 ET D3 ;
*
*-------- Utilisation de la procedure PASAPAS ---------
*-- Les objets utilisés par la procédure PASAPAS sont -
*-------- définis dans les lignes qui suivent ---------
*
MO = MODL SU MECANIQUE ELASTIQUE PLASTIQUE
     DRUCKER_PRAGER QUA4 ;

KLL= 3.56 ;
ETT= 0.441;
MUU= 1.02 ;
MAT1 = MATR MO YOUN 40000  NU 0.21
ALFA 0.441 BETA 1.02 K 4.1 GAMM 0.441 DELT 1.02 ETA
ETT  MU MUU KL KLL  H 0. ;
*
*------- Création d'une relation linéaire entre -------
*----------- le déplacement et le temps ---------------
* Remarque : au temps T, le déplacement vaut F(T)*D .
*
LI1 = PROG 0. 1. 2. 3. 4. 5.  ;
LI2 = PROG 0. 1. 2. 3. 4. 5.  ;
EV = EVOL MANU T LI1 F(T) LI2 ;
CHA1 = CHAR DIMP  DEP EV ;
*
*------- Création de la liste des pas de calcul -------
*
LIS1 = PROG 0. 1. 2.;
TAB1 = TABLE ;
TAB1.'CARACTERISTIQUES' = MAT1;
TAB1.'CHARGEMENT' = CHA1;
TAB1.'MODELE' = MO;
TAB1.'TEMPS_CALCULES' = LIS1;
TAB1.'BLOCAGES_MECANIQUES' = CL;
*
*---------- Calcul par la procedure PASAPAS -----------
*
PASAPAS TAB1;
*
*    recherche des contraintes resultats
*    par la procedure PECHE
*
SIG1 = PECHE TAB1 CONTRAINTES 1.;
SIG2 = PECHE TAB1 CONTRAINTES 2.;
*
*  extraction d'une valeur d'une composante
*  en un point de gauss
*
SXX = EXTR SIG2 'SMXX' 1 1 1 ;
syy = EXTR sig2 'SMYY' 1 1 1 ;
*
*   calcul de la valeur th{orique
*
VALTH = KLL / ( ( 2. * ETT ) +  MUU );
SAUT PAGE ; SAUT 2 LIGN ;
*
MESS '        CONTRAINTE THEORIQUE
      CONTRAINTES CALCULEES   ';
SAUT 1 LIGN ;
MESS '           SIG = '  VALTH  '
      SMXX=' SXX '  SMYY=' syy ;
*
TEMPS ;
*             CODE FONCTIONNEMENT
RESI1=ABS((SXX-VALTH)/VALTH);
RESI2=ABS((SYY-VALTH)/VALTH);
SI((RESI1 <EG 5E-2) ET (RESI2 <EG 5E-2));
    ERRE  0;
SINO;
    ERRE  5;
FINSI;
FIN;
```

Test plas5 **Comments**

1. MODEL AND CHARACTERISTICS

```
MO = MODE SU MECANIQUE ELASTIQUE PLASTIQUE
     DRUCKER_PRAGER  QUA4 ;
KLL= 3.56 ;   ETT= 0.441; MUU= 1.02 ;
MAT1 = MATE MO YOUN 40000  NU 0.21
ALFA 0.441 BETA 1.02 K 4.1 GAMM 0.441
DELT 1.02 ETA  ETT  MU MUU
KL KLL  H 0. ;
```

The DRUCKER PRAGER model enables the user to perform a calculation with non-associated yield and with stress hardening. The equations of this model are as follows :
initial criterion :

$$ALFA * Tr(S) + BETA * Seq = K$$

ultime criterion :

$$ETA * Tr(S) + MU * Seq = KL$$

hardening :

$$dK = H * dp \quad \text{(H in algebraic value)}$$

Flow potential :

$$GAMM * Tr(S) + DELT * Seq$$

For this model the parameters to define are : ALFA, BETA, K, ETA, MU, KL, H, GAMM, and DE.
S is stress tensor
Seq is Von Mises equivalent stress
p is cumulated equivalent plastic strain

2. FIND THE STRESSES WITH PECHE PROCEDURE

```
SIG1 = PECHE TAB1 CONTRAINTES 1.;
SIG2 = PECHE TAB1 CONTRAINTES 2.;
```

The PECHE procedure enables the user to retreive for a given time the results of a calculation carried out using the PASAPAS procedure. Contents :

TAB1 : table used in PASAPAS (TABLE type)

CONTRAINTES : key word (MOT type) corresponding to the requested stress index

1. (or 2.) : time (FLOTTANT type) for which the results (stresses) are requested.
By default, the results of the last calculated time will be retreived.

SIG1 (or SIG2) : generated stress field (CHPOINT type)

3.5 Test plas6 Description sheet

Test name plas6

Calculation type MECHANICS ELASTIC PLASTIC - PLANE STRESS **Finite element type** QUA4

Topic Rectangle subjected to pure bending.

A rectangle is embedded from two adjacent sides. Linear imposed displacements are applied to one side. Calculations are performed for plane stresses and with a pure plastic model. **Goal** Calculate the pure

bending moment along the side where the displacements are applied.

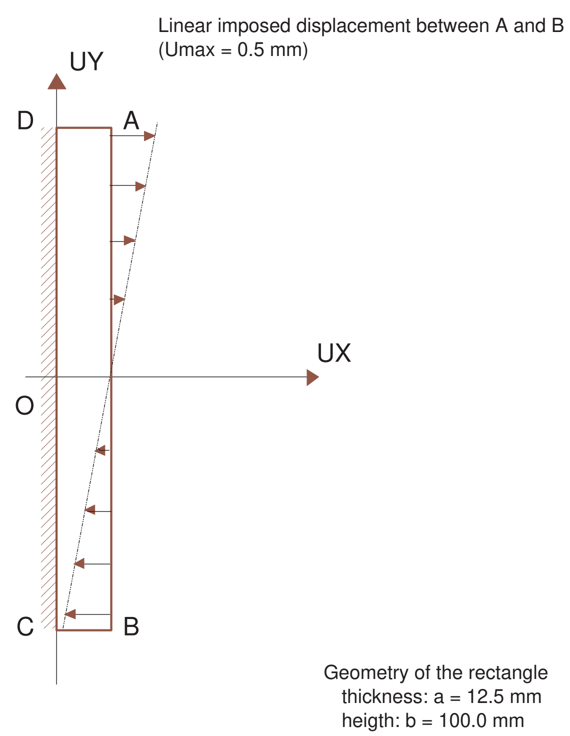
Reference CASTEM R. Hill, Mathematical theory of plasticity, Oxford, Oxford Clavendon Press, 1950. Test V.P.C.S. numero SSNP11/89

Version 97' customer version

Model description

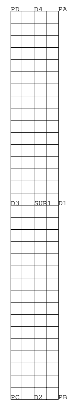
Elastic and pure plastic material

YOUNG	=	$2\ 10^{11}$ Pa
NU	=	0.3
SIGY	=	483 MPa

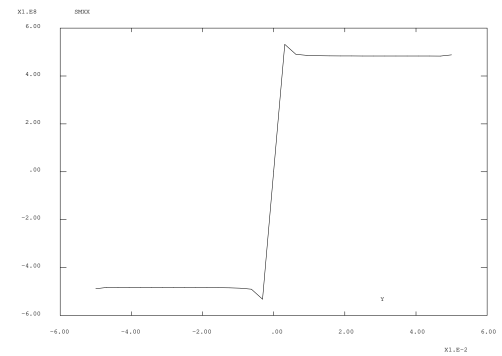


Test plas6
Results

CASTEM FIGURES

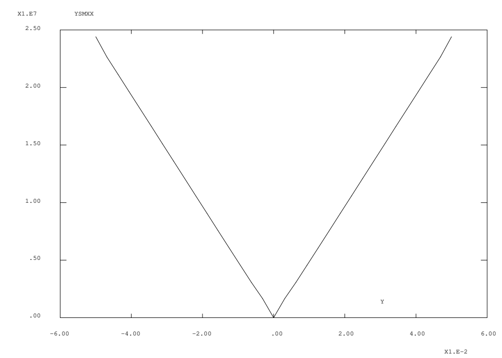


PLAS6 : MESHING



STRESS SMXX

PLAS6: STRESS SMXX



PRODUCT Y * SMXX

PLAS6: DEFORMED

```
*          Test Plas6.dgibi: Jeux de données        *
*          --------------------------------         *
*                                                   *
* CAS TEST DU 91/07/23    PROVENANCE : BIRET
SAUT PAGE;
*
*******************************************************
*                                                   *
*  TEST PLAS6                                       *
*                                                   *
*  RECTANGLE EN FLEXION PURE (CONTRAINTES PLANES,   *
*  PLASTICITE PARFAITE)                             *
*                                                   *
*  Test V.P.C.S. numero SSNP11/89                   *
*                                                   *
*  References : R. Hill, Mathematical theory of     *
*  plasticity, Oxford, Oxford Clavendon Press, 1950. *
*                                                   *
*  Description :                                    *
*  -----------                                      *
*                                                   *
*          axe y                                    *
*         D |      A                                *
*           |----- ----  UA    epaisseur a = 12.5 mm *
*           |     |   /        hauteur b = 100.0 mm *
*           |     |  /                              *
*           |     | /                               *
*           |     |/                                *
*         O |-----/-----> axe x                     *
*           |    /|                                 *
*           |   / |                                 *
*           |  /  |                                 *
*           | /   |                                 *
*            -----                                  *
*          C  UB   B                                *
*                                                   *
*  Chargement : Deplacement impose variant          *
*  lineairement de UA a -UA entre A et B,           *
*  UA croissant jusqu'a : UAmax = 0.5 mm .          *
*                                                   *
*  Conditions aux limites :                         *
*       sur CD : u = 0.                             *
*       au point O : v = 0.                         *
*                                                   *
*  Proprietes des materiaux :                       *
*       Materiau elastique, parfaitement plastique  *
*        E = 210 000 MPa , NU = 0.3 , SIGY = 483 MPa *
*                                                   *
*******************************************************
*
OPTI ECHO 1;
*
OPTI DIME 2;
OPTI MODE PLAN CONT;
OPTI ELEM QUA4;
OPTI ISOV LIGNE;
*
GRAPH = 'N';
GRAPH2 = 'N';
*
TEMPS;
*******************************************************
*                 geometrie : maillage              *
*******************************************************
*
PA = 12.5E-3  50.E-3;
PB = 12.5E-3 -50.E-3;
PC =  0.     -50.E-3;
PD =  0.      50.E-3;
*
NB1 = 32; NB2 = 4;
D1 = PA DROI NB1 PB;
D2 = PB DROI NB2 PC;
D3 = PC DROI NB1 PD;
D4 = PD DROI NB2 PA;
*
SUR1 = DALL D1 D2 D3 D4 PLAN;
*
SI (NEG GRAPH 'N');
   TITR 'PLAS6 : MAILLAGE';
   TRAC QUAL SUR1;
FINSI;
*

NBNO1 = NBNO SUR1;
NBEL1 = NBEL SUR1;
*
*******************************************************
*          calcul du modele,caracteristiques         *
*******************************************************
*
MOD1 = MODL SUR1 MECANIQUE ELASTIQUE PLASTIQUE
       PARFAIT QUA4;
MAT1 = MATR MOD1 YOUN 2.1E11 NU 0.3 SIGY 483.E6;
*
*******************************************************
*                 conditions aux limites             *
*******************************************************
*
CL1 = BLOQ UX D3;
CL2 = BLOQ UY (SUR1 POIN PROC (0. 0.));
CL3 = BLOQ UX D1;
CL = CL1 ET CL2 ET CL3;
*
* Creation du chpoint de deplacement impose variable :
*   deplacement impose variant lineairement de u a -u
*   entre A et B, u croissant jusqu'a 0.5 mm .
*
I = 0.;
LISD1 = PROG 1.;
REPETER BLOC10 NB1;
  I = I + 1.;
  YY = 1. - (2. * (I / NB1));
  LISD1 = LISD1 ET (PROG YY);
FIN BLOC10;
*
LISD1 = LISD1 * 1.E-3;
CHPO1 = MANU CHPO D1 1 UX LISD1;
*
CHA1 = DEPI CL3 CHPO1;
*
*******************************************************
*      objets relatifs a la procedure PASAPAS        *
*******************************************************
*
TAB1 = TABLE;
*
*On prend comme liste des pseudo-temps le deplacement u
*
LREEL1= PROG 0.02875 0.05 0.1 0.15 0.2 0.3 0.4 0.5 1.0;
*
* Chargement
*
EVOL1 = EVOL MANU UA LREEL1 COEFF LREEL1;
CHAR1 = CHAR DIMP CHA1 EVOL1;
*
```

```
*******************************************************
*                     resolution                      *
*******************************************************
*
TAB1.'CARACTERISTIQUES' = MAT1;
TAB1.'CHARGEMENT' = CHAR1;
TAB1.'MODELE' = MOD1;
TAB1.'TEMPS_CALCULES' = LREEL1;
TAB1.'BLOCAGES_MECANIQUES' = CL;
PASAPAS TAB1;
*
*******************************************************
*               extraction des resultats              *
*******************************************************
*
* Initialisation
*
MOM1 = TABLE;
LISD2 = LISD1 * 50.;
MOM2 = TABLE;
ERG = TABLE;
*
MOM2.1 = 805.0; MOM2.2 = 1074.;
MOM2.3 = 1174.; MOM2.4 = 1193.;
MOM2.5 = 1199.; MOM2.6 = 1204.;
MOM2.7 = 1205.; MOM2.8 = 1206.;
MOM2.9 = 1207.5;
*
*Boucle pour chaque pas de chargement : fonction de UA.
*
I = 0;
REPETER BLOC1 9;
  I = I + 1;
  CONT1 = TAB1.CONTRAINTES.I;
  DEPL1 = TAB1.DEPLACEMENTS.I;
*
* Passage du champs de contrainte en chpoints
*   (utilisation des nouveaux chamelems)
*

  CHPO10 = CHAN CHPO MOD1 CONT1;
*
*       Trace facultatif de la repartition
*             de la contrainte SMXX
*
SI (I EGA 9);
  SI (NEG GRAPH2 'N');
    TITR 'PLAS6 : CONTRAINTE SMXX';
    BB = EXCO 'SMXX' CONT1;
    TRAC BB SUR1;
  FINSI;
FINSI;
*
*        Trace facultatif de la deformee
*         (verification du chargement)
*
SI (I EGA 9);
  SI (NEG GRAPH2 'N');
     TITR 'PLAS6 : DEFORMEE';
     DEF0 = DEFO SUR1 DEPL1 0.;
     DEF1 = DEFO SUR1 DEPL1 2. VERT;
     TRACE SUR1 (DEF0 ET DEF1);
  FINSI;
FINSI;
*
*   Calcul du moment de flexion du aux contraintes
* smxx s'exercant le long de AB.
*
  LIST2 = PROG;
  J = 0;
  NB = NB1 + 1;
  REPETER BLOC4 NB;
     J = J + 1;
     D = EXTR LISD2 J;
     POI1 = SUR1 POIN PROC (12.5E-3 D);
     SM = EXTRA CHPO10 SMXX POI1;
     LIST2 = LIST2 ET (PROG SM);
  FIN BLOC4;
*
  LIST3 = LIST2 * LISD2;
  TITR 'CONTRAINTE SMXX';
  ESMXX = EVOL MANU Y LISD2 SMXX LIST2;
  TITR 'PRODUIT Y * SMXX';
  EYSMXX = EVOL MANU Y LISD2 YSMXX LIST3;
*
SI (I EGA 9);
  SI (NEG GRAPH2 'N');
    DESSIN ESMXX;
    DESSIN EYSMXX;
  FINSI;
FINSI;
  MOM1.I = SOMME EYSMXX;
  MOM1.I = EXTR MOM1.I 1;

*
  MOM1.I = -1.E-3 * MOM1.I;
*
  ERG.I = 100. * (ABS ((MOM1.I - MOM2.I) / MOM2.I));
*
FIN BLOC1;
*
*******************************************************
*             affichage des resultats                 *
*******************************************************
*
MESS 'RESULTATS : PLAS6';
MESS '-------------------';
SAUT LIGN;
*
MESS 'Moment de flexion du aux contraintes SMXX
le long de AB.';
MESS '--------------------------------------------------------';
SAUT LIGN;
*
MESS '  UAmax = 0.02875 mm :';
MESS '  ------------------';
MESS '    Moment theorique :' MOM1.1 'N.m';
MESS '    Moment calcule   :' MOM2.1 'N.m';
MESS '    Soit un ecart de :' ERG.1 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.05 mm :';
MESS '  ---------------  ';
MESS '    Moment theorique :' MOM1.2 'N.m';
MESS '    Moment calcule   :' MOM2.2 'N.m';
MESS '    Soit un ecart de :' ERG.2 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.1 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM1.3 'N.m';
MESS '    Moment calcule   :' MOM2.3 'N.m';
MESS '    Soit un ecart de :' ERG.3 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.15 mm :';
```

```
MESS '  ---------------';
MESS '    Moment theorique :' MOM1.4 'N.m';
MESS '    Moment calcule   :' MOM2.4 'N.m';
MESS '    Soit un ecart de :' ERG.4 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.2 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM1.5 'N.m';
MESS '    Moment calcule   :' MOM2.5 'N.m';
MESS '    Soit un ecart de :' ERG.5 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.3 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM1.6 'N.m';
MESS '    Moment calcule   :' MOM2.6 'N.m';
MESS '    Soit un ecart de :' ERG.6 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.4 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM1.7 'N.m';
MESS '    Moment calcule   :' MOM2.7 'N.m';
MESS '    Soit un ecart de :' ERG.7 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.5 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM1.8 'N.m';
MESS '    Moment calcule   :' MOM2.8 'N.m';
MESS '    Soit un ecart de :' ERG.8 '%';
SAUT LIGN;
*
MESS '  UAmax = 1.0 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM1.9 'N.m';
MESS '    Moment calcule   :' MOM2.9 'N.m';
MESS '    Soit un ecart de :' ERG.9 '%';
SAUT LIGN;
*
MESS 'Donnees sur le maillage';
MESS '-----------------------';
MESS '    Nombre d elements :' NBEL1 'QUA4';
MESS '    Nombre de noeuds  :' NBNO1 ;
SAUT LIGN;
*
*******************************************************
*                   code fonctionnement               *
*******************************************************
*
ERGMAX = MAXI (PROG ERG.1 ERG.2 ERG.3 ERG.4 ERG.5 ERG.6
                   ERG.7 ERG.8 ERG.9);
*
SI (ERGMAX <EG 5.);
  ERRE 0;
SINON;
  ERRE 5;
FINSI;
SAUT LIGN;
*
TEMPS;
*
FIN;
*
```

3.6 Test plas7 Description sheet

Test name plas7

Calculation type MECHANICS ELASTIC PLASTIC - PLANE STRAIN **Finite element type** QUA44

Topic Rectangle subjected to pure bending.

A rectangle is embedded from two adjacent sides. Linear imposed displacements are applied to one side. Calculations are made for plan strains and with a pure plastic model. **Goal** Calculate the pure bending

moment along the side where the displacements are applied.

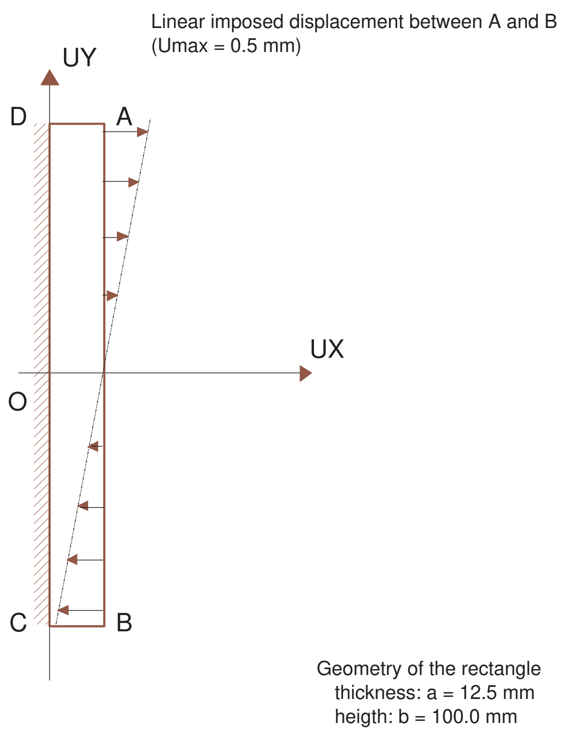
Reference CASTEM R. Hill, Mathematical theory of plasticity, Oxford, Oxford Clavendon Press, 1950. Test V.P.C.S. numero SSNP11/89

Version 97' customer version

Model description

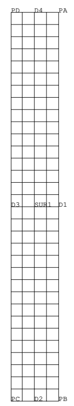
Elastic and pure plastic material

YOUNG	=	$2\ 10^{11}$ Pa
NU	=	0.3
SIGY	=	483 MPa

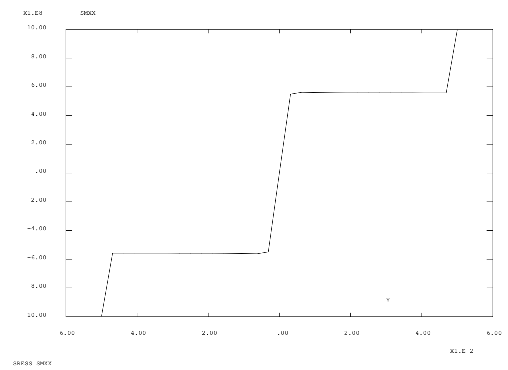


Test plas7
Results

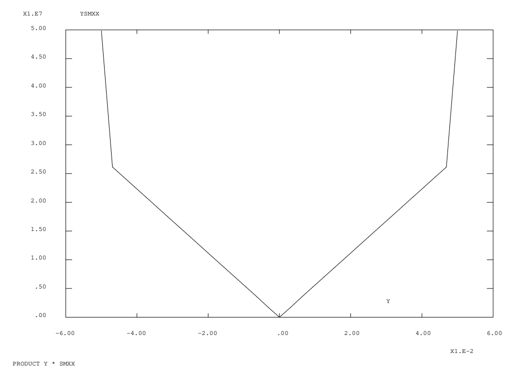
CASTEM FIGURES



PLAS7: MESH



PLAS7: STRESS SMXX



PLAS7: DEFORMED

```
*          Test Plas7.dgibi: Jeux de données        *
*          --------------------------------         *
*                                                   *
* CAS TEST DU 91/06/13     PROVENANCE : TEST
SAUT PAGE;
*
*******************************************************
*                                                     *
*  TEST PLAS7                                         *
*                                                     *
*  RECTANGLE EN FLEXION PURE (DEFORMATIONS PLANES,    *
*  PLASTICITE PARFAITE)                               *
*                                                     *
*  Test V.P.C.S. numero SSNP11/89                     *
*                                                     *
*  References : R. Hill, Mathematical theory of       *
*  plasticity, Oxford, Oxford Clavendon Press, 1950.  *
*                                                     *
*  Description :                                      *
*  -----------                                        *
*                                                     *
*          axe y                                      *
*         D |       A                                 *
*           |----- ----  UA    epaisseur a = 12.5 mm *
*           |     |   /        hauteur b = 100.0 mm  *
*           |     |  /                                *
*           |     | /                                 *
*           |     |/                                  *
*         O |-----/-----> axe x                       *
*           |    /|                                   *
*           |   / |                                   *
*           |  /  |                                   *
*           | /   |                                   *
*            -----                                    *
*          C  UB   B                                  *
*                                                     *
*  Chargement : Deplacement impose variant            *
*  lineairement de UA a -UA entre A et B,             *
*  UA croissant jusqu'a : UAmax = 0.5 mm .            *
*                                                     *
*  Conditions aux limites :                           *
*       sur CD : u = 0.                               *
*       au point O : v = 0.                           *
*                                                     *
*  Proprietes des materiaux :                         *
*       Materiau elastique, parfaitement plastique    *
*        E = 210 000 MPa , NU = 0.3 , SIGY = 483 MPa  *
*                                                     *
*******************************************************

*
OPTI ECHO 1;
*
OPTI DIME 2;
OPTI MODE PLAN DEFO;
OPTI ELEM QUA4;
OPTION ISOV LIGNE;
*
GRAPH = 'N';
GRAPH2 = 'N';
*
TEMPS;
*+++++++++++++++++++++++++++++++++++++++++++++++++++++*
*                 geometrie : maillage                *
*+++++++++++++++++++++++++++++++++++++++++++++++++++++*
*
PA = 12.5E-3  50.E-3;
PB = 12.5E-3 -50.E-3;
PC =  0.     -50.E-3;
PD =  0.      50.E-3;
*
NB1 = 32; NB2 = 4;
D1 = PA DROI NB1 PB;
D2 = PB DROI NB2 PC;
D3 = PC DROI NB1 PD;
D4 = PD DROI NB2 PA;
*
SUR1 = DALL D1 D2 D3 D4 PLAN;
*
SI (NEG GRAPH 'N');
   TITR 'PLAS7 : MAILLAGE';
   TRAC QUAL SUR1;
FINSI;
*
NBNO1 = NBNO SUR1;
NBEL1 = NBEL SUR1;
*
*+++++++++++++++++++++++++++++++++++++++++++++++++++++*
*     definition du modele, des caracteristiques      *
*+++++++++++++++++++++++++++++++++++++++++++++++++++++*
*
MOD1 = MODL SUR1 MECANIQUE ELASTIQUE PLASTIQUE
       PARFAIT QUA4;
MAT1 = MATR MOD1 YOUN 2.1E11 NU 0.3 SIGY 483.E6;
*
*+++++++++++++++++++++++++++++++++++++++++++++++++++++*
*                  conditions aux limites             *
*+++++++++++++++++++++++++++++++++++++++++++++++++++++*
*
CL1 = BLOQ UX D3;
CL2 = BLOQ UY (SUR1 POIN PROC (0. 0.));
CL3 = BLOQ UX D1;
CL = CL1 ET CL2 ET CL3;
*
* Creation du chpoint de deplacement impose variable :
*   deplacement impose variant lineairement de u a -u
*   entre A et B, u croissant jusqu'a 0.5 mm .
*
I = 0.;
LISD1 = PROG 1.;
REPETER BLOC10 NB1;
  I = I + 1.;
  YY = 1. - (2. * (I / NB1));
  LISD1 = LISD1 ET (PROG YY);
FIN BLOC10;
*
LISD1 = LISD1 * 1.E-3;
CHPO1 = MANU CHPO D1 1 UX LISD1;
*
CHA1 = DEPI CL3 CHPO1;
*
*+++++++++++++++++++++++++++++++++++++++++++++++++++++*
*      objets relatifs a la procedure PASAPAS         *
*+++++++++++++++++++++++++++++++++++++++++++++++++++++*
*
TAB1 = TABLE;
*
*       On prend comme liste des pseudo-temps
*                 le deplacement u.
*
LREEL1 = PROG 0.029435 0.1 0.2 0.3 0.4 0.5 1.0;
*
* Chargement
*
EVOL1 = EVOL MANU UA LREEL1 COEFF LREEL1;
CHAR1 = CHAR DIMP CHA1 EVOL1;
```

```
*
*******************************************************
*                    resolution                       *
*******************************************************
*
TAB1.'CARACTERISTIQUES' = MAT1;
TAB1.'CHARGEMENT' = CHAR1;
TAB1.'MODELE' = MOD1;
TAB1.'TEMPS_CALCULES' = LREEL1;
TAB1.'BLOCAGES_MECANIQUES' = CL;
PASAPAS TAB1;
*
*******************************************************
*              extraction des resultats               *
*******************************************************
*
* Initialisation
*
MOM1 = TABLE;
LISD2 = LISD1 * 50.;
MOM2 = TABLE;
ERG = TABLE;
*
MOM2.1 = 90.57;
MOM2.2 = 135.2;
MOM2.3 = 138.4;
MOM2.4 = 139.0;
MOM2.5 = 139.2;
MOM2.6 = 139.3;
MOM2.7 = 139.4;
*
MOM2.8 = 543.4;
MOM2.9 = 557.7;
*
*Boucle pour chaque pas de chargement : fonction de UA.
*
I = 0;
REPETER BLOC1 7;
  I = I + 1;
  CONT1 = TAB1.CONTRAINTES.I;
  DEPL1 = TAB1.DEPLACEMENTS.I;
  CHPO10 = CHAN CHPO MOD1 CONT1;
*
*          Trace facultatif de la repartition
*                de la contrainte SMXX
*
SI (I EGA 7);

  SI (NEG GRAPH2 'N');
    TITR 'PLAS7 : CONTRAINTE SMXX';
    BB = EXCO 'SMXX' CONT1;
    TRAC BB SUR1;
  FINSI;
FINSI;
*
*          Trace facultatif de la deformee
*            (verification du chargement)
*
SI (I EGA 7);
 SI (NEG GRAPH2 'N');
     TITR 'PLAS7 : DEFORMEE';
     DEF0 = DEFO SUR1 DEPL1 0.;
     DEF1 = DEFO SUR1 DEPL1 2. VERT;
     TRACE SUR1 (DEF0 ET DEF1);
  FINSI;
FINSI;
*
*    Calcul du moment de flexion du aux contraintes
* smxx s'exercant le long de AB.
*
  LIST2 = PROG;
  J = 0;
  NB = NB1 + 1;
  REPETER BLOC4 NB;
     J = J + 1;
     D = EXTR LISD2 J;
     POI1 = SUR1 POIN PROC (12.5E-3 D);
     SM = EXTRA CHPO10 SMXX POI1;
     LIST2 = LIST2 ET (PROG SM);
  FIN BLOC4;
*
  LIST3 = LIST2 * LISD2;
  TITR 'CONTRAINTE SMXX';
  ESMXX = EVOL MANU Y LISD2 SMXX LIST2;
  TITR 'PRODUIT Y * SMXX';
  EYSMXX = EVOL MANU Y LISD2 YSMXX LIST3;
*
SI (I EGA 7);

  SI (NEG GRAPH2 'N');
    DESSIN ESMXX;
    DESSIN EYSMXX;
  FINSI;

FINSI;

  MOM1.I = SOMME EYSMXX;
  MOM1.I = EXTR MOM1.I 1;

*
  MOM1.I = -1.E-4 * MOM1.I;
*
* Calcul de l'ecart entre theorie et calcul.
*
  ERG.I = 100. * (ABS ((MOM1.I - MOM2.I) / MOM2.I));
*
*       Extraction de la contrainte maximale SMXX
*               pour les pas 1 et 7.
*
SI (I EGA 1);
 MOM1.8 = EXTR CHPO10 SMXX PA;
 MOM1.8 = MOM1.8 / 1.E6;
 ERG.8 = 100. * (ABS ((MOM1.8 - MOM2.8) / MOM2.8));
FINSI;
*
SI (I EGA 7);
 MOM1.9 = EXTR CHPO10 SMXX (D1 POIN 2);
 MOM1.9 = MOM1.9 / 1.E6;
 ERG.9 = 100. * (ABS ((MOM1.9 - MOM2.9) / MOM2.9));
FINSI;
*
FIN BLOC1;
*
*******************************************************
*             affichage des resultats                 *
*******************************************************
*
MESS 'RESULTATS : PLAS7';
MESS '-------------------';
SAUT LIGN;
*
MESS 'Moment de flexion du aux contraintes SMXX
le long de AB.';
MESS '------------------------------------------------------';
SAUT LIGN;
*
```

```
MESS '  UAmax = 0.029435 mm :';
MESS '  ------------------- ';
MESS '    Moment theorique :' MOM2.1 'kN.m';
MESS '    Moment calcule   :' MOM1.1 'kN.m';
MESS '    Soit un ecart de :' ERG.1 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.1 mm :';
MESS '  --------------  ';
MESS '    Moment theorique :' MOM2.2 'kN.m';
MESS '    Moment calcule   :' MOM1.2 'kN.m';
MESS '    Soit un ecart de :' ERG.2 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.2 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM2.3 'kN.m';
MESS '    Moment calcule   :' MOM1.3 'kN.m';
MESS '    Soit un ecart de :' ERG.3 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.3 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM2.4 'kN.m';
MESS '    Moment calcule   :' MOM1.4 'kN.m';
MESS '    Soit un ecart de :' ERG.4 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.4 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM2.5 'kN.m';
MESS '    Moment calcule   :' MOM1.5 'kN.m';
MESS '    Soit un ecart de :' ERG.5 '%';
SAUT LIGN;
*
MESS '  UAmax = 0.5 mm :';
MESS '  --------------';
MESS '    Moment theorique :' MOM2.6 'kN.m';
MESS '    Moment calcule   :' MOM1.6 'kN.m';
MESS '    Soit un ecart de :' ERG.6 '%';
SAUT LIGN;
*
MESS '  UAmax = 1.0 mm :';
MESS '  ---------------';
MESS '    Moment theorique :' MOM2.7 'kN.m';
MESS '    Moment calcule   :' MOM1.7 'kN.m';
MESS '    Soit un ecart de :' ERG.7 '%';
SAUT LIGN;
*
MESS 'Contrainte maximale smxx :';
MESS '------------------------ :';
SAUT LIGN;
MESS '  UAmax = 0.029435 mm';
MESS '  -------------------';
MESS '    Valeur theorique :' MOM2.8 'MPa';
MESS '    Valeur calculee  :' MOM1.8 'MPa';
MESS '    Soit un ecart de :' ERG.8 '%';
SAUT LIGN;
MESS '  UAmax = 1.0 mm';
MESS '  --------------';
MESS '    Valeur theorique :' MOM2.9 'MPa';
MESS '    Valeur calculee  :' MOM1.9 'MPa';
MESS '    Soit un ecart de :' ERG.9 '%';
SAUT LIGN;
*
MESS 'Donnees sur le maillage';
MESS '-----------------------';
MESS '    Nombre d elements :' NBEL1 'QUA4';
MESS '    Nombre de noeuds  :' NBNO1 ;
SAUT LIGN;
*
*******************************************************
*                   code fonctionnement               *
*******************************************************
*
ERGMAX = MAXI (PROG ERG.1 ERG.2 ERG.3 ERG.4 ERG.5 ERG.6
                   ERG.7 ERG.8 ERG.9);
*
SI (ERGMAX <EG 5.1);
  ERRE 0;
SINON;
  ERRE 5;
FINSI;
SAUT LIGN;
*
TEMPS;
*
FIN;
*
```

3.7 Test plas8 Description sheet

Test name plas8

Calculation type MECHANICS ELASTIC PLASTIC

Finite element type TRI3

Topic Punctual strength as a function of the deflection..

A beam is embedded at one side and at the free side we impose a vertical displacement (e.g. a deflection).

Goal Calculate the evolution of the reaction strength as a function of the deflection. According to the pure elastoplastic law the strength should converge to an asymptotic value given by the following relation:

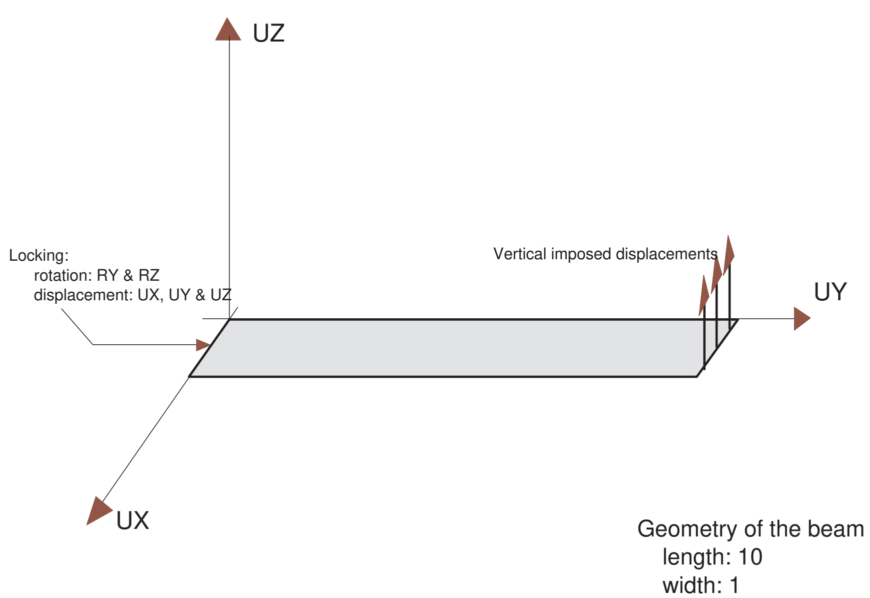
FU = (SIGY*H2/4)/L
Where:
SIGY = stress at the yield point
H = thickness of the beam
L = length of the beam

Version 97' customer version

Model description

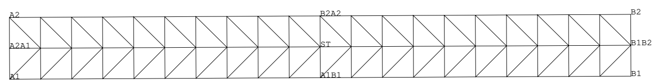
Elastic and pure plastic material

YOUNG	=	$2.1\ 10^{11}$ Pa
NU	=	0.
SIGY	=	100. MPa

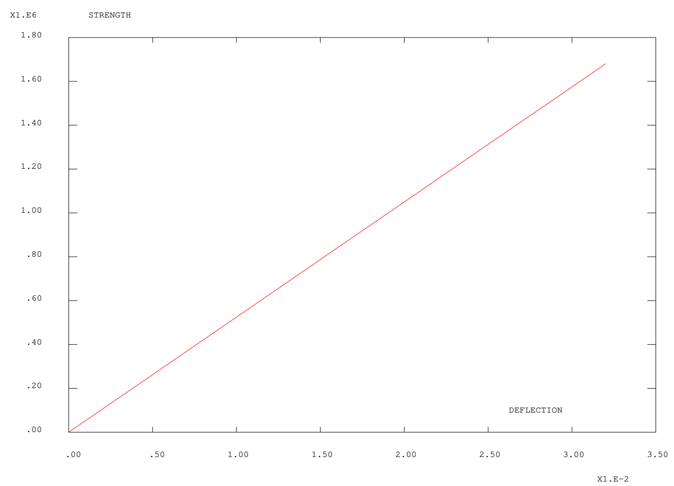


Test plas8 **Results**

CASTEM FIGURES



PUNCTUAL STRENGTH AS A FUNCTION OF DEFLECTION



PUNCTUAL STRENGTH AS A FUNCTION OF DEFLECTION

```
*          Test Plas8.dgibi: Jeux de données
*          ---------------------------------
*
*
complet = faux;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
*************************************************

OPTION ECHO 0;
GRAPH='N';
*opti trac bens;
SAUT PAGE ;
*
*******************************************************
*  TEST PLAS8
*
*   CONSOLE SOUS CHARGE CONCENTREE
*
*   On impose un deplacement vertical (une fleche) sur
*   l'extremite libre d'une console. On etudie
*   l'evolution de la force de reaction fonction de la
*   fleche.La force doit tendre vers une valeur
*   asymptotique donnee par l'expression suivante
*   (si on considere la loi elastoplastique parfaite):
*
*   FU = (SIGY*H2/4)/L
*    ou  SIGY = contrainte limite elastique
*        H    = l'epaisseur de la poutre
*        L    = longueur de poutre
*
*******************************************************
TITRE 'FORCE PONCTUELLE FONCTION DE LA FLECHE  ';
OPTI DIME 3 ELEM TRI3 ECHO 0  DENS 0.5 ;
*______________________________________________________
*  GEOMETRIE
*______________________________________________________
B=  1. ;  L=10. ;
A1 = 0. 0. 0. ;A2=0. B  0.0;
B1 =  L   0. 0. ; B2 = L  B 0.0 ;
nel = 20;
A2A1=DROIT 2 A2 A1;A1B1=DROIT nel A1 B1 ;
B1B2=DROIT 2 B1 B2;B2A2=DROIT nel B2 A2 ;
ST=DALLER A2A1 A1B1 B1B2 B2A2 PLAN ;
OEIL = 0 0 1000 ;
SI (NEG GRAPH 'N') ;
  TRACE OEIL ST  ;
FINSI ;
*______________________________________________________
*  INITIALISATION
*______________________________________________________
XABS=PROG ;
FONC1=PROG;
EP=L/10 ;
*______________________________________________________
*   MODELE
*______________________________________________________
MOD1=MODL ST MECANIQUE ELASTIQUE PLASTIQUE  PARFAIT
 DKT  'INTEGRE' 7 ;
*______________________________________________________
*  MATERIAUX
*______________________________________________________
E=2.1E11 ; SY= 100.E6;
MAT1=MATR MOD1 YOUN E NU 0. SIGY SY  EPAI EP ALFA 1. ;
*______________________________________________________
*  CONDITIONS AUX LIMITES ET RIGIDITE
*______________________________________________________
* RI1=RIGI MOD1 MAT1 ;
* CL1=BLOQ UX UY UZ  RY  RZ A2A1 ;
* CLDEP=BLOQ UZ B1B2 ;
CL = CL1 ET CLDEP;
RIT1 = RI1 ET  CL;
*______________________________________________________
*   DEPLACEMENT IMPOSE
*______________________________________________________
 FDEP=DEPI CLDEP  0.2;
*______________________________________________________
*  PREPARATION POUR PASAPAS
*______________________________________________________
*
*
LI1 = PROG 0. 1. ;
LI2 = PROG 0. 1. ;
EV = EVOL MANU T LI1 F(T) LI2 ;
CHA1 = CHAR DIMP FDEP  EV ;
*
si complet;
 LIS1 = PROG PAS 0.03333 0.1 PAS 0.02 0.3 PAS 0.01 0.4;
sinon;
 LIS1 = PROG  PAS 0.03333 0.1 0.12 0.14 0.16 ;
finsi;
TAB1 = TABLE ;
TAB1.'CARACTERISTIQUES' = MAT1;
TAB1.'CHARGEMENT' = CHA1;
TAB1.'MODELE' = MOD1;
TAB1.'TEMPS_CALCULES' = LIS1;
TAB1.'BLOCAGES_MECANIQUES' = CL;
*______________________________________________________
* CALCUL
*______________________________________________________
PASAPAS TAB1;
TEMPS ;
*
*______________________________________________________
* POST-TRAITEMENT
*______________________________________________________
ITER=0 ;
opti echo 1;
NB=DIME LIS1 ;
XABS=INSER XABS  1 (0.0) ;
FONC1=INSER FONC1 1 (0.0);
REPETE BOUC1 NB ;
ITER=ITER+1 ;
ITER1=ITER+1 ;
TT=EXTR LIS1 ITER ;
DEP=PECHE TAB1 'DEPLACEMENTS' TT ;
FLECH=EXTR DEP UZ B1;
FF=REAC DEP RIT1 ;
FFF=RESUL(REDU FF B1B2)  ;
FORC=EXTR FFF 'FZ' B1 ;
XABS=INSER XABS  ITER1 FLECH ;
FONC1=INSER FONC1 ITER1 FORC;
FIN BOUC1 ;
*______________________________________________________
* ON TRACE LA COURBE FORCE-FLECHE
*______________________________________________________
EV1=EVOL ROUGE MANU 'FLECHE' XABS 'FORCE' FONC1 ;
SI(NEG GRAPH 'N');
 DESSIN EV1 ;
FINSI ;
TEMPS ;
*______________________________________________________
* CALCUL ANALYTIQUE DE LA VALEUR ASYMPTOTIQUE
*______________________________________________________
si complet;
```

```
  VAL1=SY*(EP**2.) ;FU=(VAL1/4)/L ;
sinon;
 fu = 1.679E+06;
finsi;
*
*------------------------------------------------------
*            CODE FONCTIONNEMENT
*------------------------------------------------------
*
FC=RESUL(REDU FF B1B2) ;
FCC=EXTR FC 'FZ' B1 ;
RESI=ABS((FCC-FU)/FU) ;
SI (RESI <EG 5E-2);
    ERRE  0 ;
SINO;
    ERRE  5 ;
FINSI;
FIN ;
```

3.8 Test plas9 Description sheet

Test name plas9

Calculation type MECHANICS ELASTIC PLASTIC - DRUCKER-PRAGER'S MATERIAL

Finite element type CUB8

Topic Unidirectionnal tensile-test on a parallelipiped.

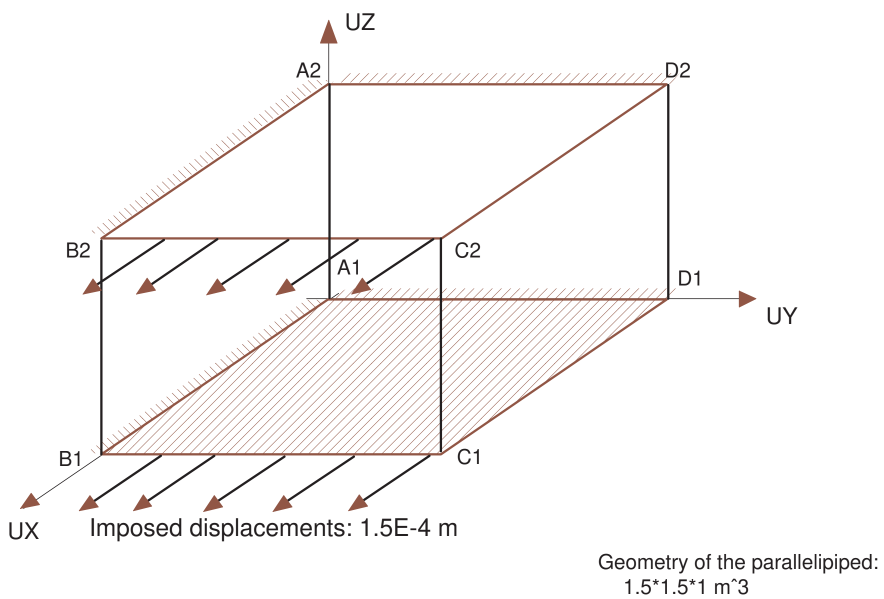
An isostatic embedded parallelipiped is subjected to an imposed displacement on one of its faces. It follows to to Drucker-Prager model (negative strain-hardening)

Goal Compare the curves of unidirectionnal behavior with the exact solution.

Version 97' customer version

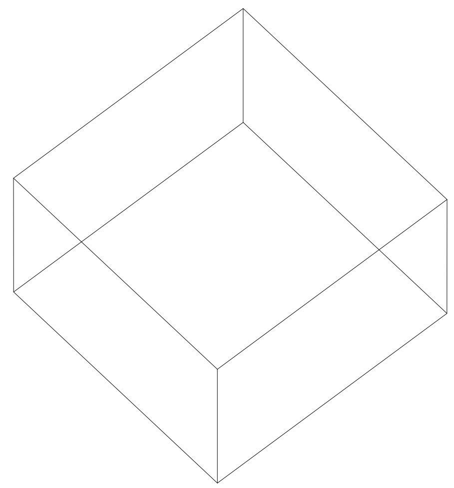
Model description

	Elastoplastic Mechanic Drucker-Prager	Elastic mechanic
YOUNG	40000	2.D-3
NU	0.21	0.

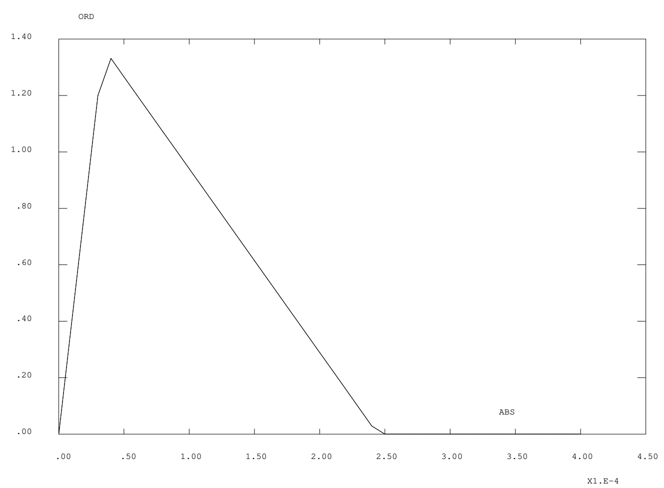


Test plas9 **Results**

CASTEM FIGURES



PARALLELEPIPEDE EN TRACTION UNIAXIALE



PARALLELEPIPEDE EN TRACTION UNIAXIALE

```
*          Test Plas9.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0;
GRAPH='N';
SAUT PAGE ;
*
*****************************************************
*  TEST PLAS9
*
* Sortie du domaine élastique et phase plastique
* (comportement élasto-plastique modèle DRUCKER-PRAGER
* à écrouissage négatif).
*
* Un parallelépipède est soumis à un déplacement imposé
* sur une de ses faces et fixé de manière isostatique.
* On compare la courbe de réponse uniaxiale à la
* solution exacte.
*
*****************************************************

TITRE 'PARALLELEPIPEDE EN TRACTION UNIAXIALE ' ;
OPTION ECHO 0 ;
OPTION DIME 3 ELEM CUB8 MODE TRID     ;
*---------------------------------------------------
*          DEFINITION DE LA GEOMETRIE
*---------------------------------------------------
A1= 0. 0. 0.;
B1= 1.5 0. 0.;
C1= 1.5 1.5 0.;
D1= 0. 1.5 0.;
A2= 0. 0. 1.;
B2= 1.5 0. 1.;
C2= 1.5 1.5 1.;
D2= 0. 1.5 1.;
*
L1 = A1 DROIT 1 B1  ;
L2 = B1  DROIT 1 C1  ;
L3 = C1  DROIT 1 D1  ;
L4 = D1  DROIT 1 A1  ;
LL = L1 ET L2 ET L3 ET L4 ;
SU1= SURF LL PLAN ;
M1 = A2 DROIT 1 B2  ;
M2 = B2  DROIT 1 C2  ;
M3 = C2  DROIT 1 D2  ;
M4 = D2  DROIT 1 A2  ;
MM = M1 ET M2 ET M3 ET M4 ;
SU2= SURF MM PLAN ;
SU = SU1 VOLU  1 SU2 ;
SI(NEG GRAPH 'N');
   TRAC (888  1000 2000 )  SU ;
FINSI;
*---------------------------------------------------
*          DEFINITION DES CONDITIONS AUX LIMITES
*               ET DES DEPLACEMENTS IMPOSES
*---------------------------------------------------
CL4 = (BLOQ L4 UX) ET ( BLOQ M4 UX)   ;
CL2 = (BLOQ L2 UX) ET ( BLOQ M2 UX)  ;
CL1 = (BLOQ L1 UY) ET (BLOQ M1 UY);
CL5 = BLOQ SU1 UZ ;
CL = CL1 ET CL2 ET CL4 ET CL5 ;
*
DEP= DEPI CL2 1.5E-4 ;
*
*---------------------------------------------------
*          DEFINITION DES PROPRIETES MATERIELLES
*---------------------------------------------------
MO1= MODE SU MECANIQUE ELASTIQUE PLASTIQUE
     DRUCKER_PRAGER CONS 'CONS1' ;
MO2= MODE SU MECANIQUE ELASTIQUE CONS 'CONS2';
MO = MO1 ET  MO2 ;
VALK = 2. ;
VALALF =0.441 ;
VALBE = 1.02 ;
VALH = -10000.;
YOYO =  40000. ;
MAT1 = MATE MO1  YOUN YOYO   NU 0.21
ALFA VALALF BETA VALBE K VALK
 GAMM VALALF DELT VALBE ETA VALALF MU VALBE
KL VALK   H VALH ;
MAT2 = MATE MO2  YOUN 2.D-3 NU 0. ;
MAT = MAT1 ET MAT2 ;
LI1 = PROG 0. PAS 1. 1000. ;
LI2 = LI1 ;
EV = EVOL MANU T LI1 F(T) LI2 ;
*-----------------------------------------
*          APPEL A PASAPAS
* AU TEMPS T , LE DEPLACEMENT VAUDRA F(T)*D
*-----------------------------------------
CHA1 = CHAR DIMP DEP EV ;
LIS1 = PROG 0. PAS 0.1  4. ;
TAB1 = TABLE ;
TAB1.'CARACTERISTIQUES' = MAT;
TAB1.'CHARGEMENT' = CHA1;
TAB1.'MODELE' = MO;
TAB1.'TEMPS_CALCULES' = LIS1;
TAB1.'BLOCAGES_MECANIQUES' = CL;
TAB1.FTOL=0.000001;
PASAPAS TAB1;
*-----------------------------------------
*          DEPOUILLEMENT ET
*     CALCUL DE LA SOLUTION ANALYTIQUE
*
*   ON VERIFIE EN GROS QUE LE CALCUL EST
*   ALLE JUSQU'AU BOUT
*-----------------------------------------
*
MU = (((2. * ( VALALF ** 2)) + ( VALBE ** 2) ) ** 0.5)
 / ( VALALF + VALBE) ;
VALHMU = VALH * MU ;
 PGX = PROG 0. ;
 PGY = PROG 0. ;
 PGT = PROG 0. ;
 TT = TAB1 . CONTRAINTES ;
 NN = (DIME TT) - 1  ;
 SI  (( TAB1 . TEMPS . NN) < 3.99) ;
   ERRE 5 ;
 FINSI;
*
 REPE BOU NN;
 FF = EXTR  ( TAB1 . CONTRAINTES . (&BOU) )
      SMXX 1 1 1 ;
 EPS = EPSI MO1 ( TAB1 . DEPLACEMENTS . (&BOU) ) ;
 GG = EXTR EPS EPXX 1 1 1 ;
 ETH = VALK / (  ( VALALF + VALBE ) * YOYO ) ;
 SI ( GG < ETH ) ;
 THE = YOYO * GG  ;
 SINON;
 THE= ( VALK + ( VALHMU * GG  )) / ( (VALALF + VALBE) +
                                  ( VALHMU / YOYO ) ) ;
 SI ( THE < 0. ) ; THE = 0. ; FINSI ;
 FINSI ;
 PGX = PGX ET ( PROG GG  ) ;
 PGY = PGY ET ( PROG FF ) ;
 PGT = PGT ET ( PROG THE ) ;
 FIN BOU ;
```

```
 EV1= EVOL ROUG  MANU ABS PGX ORD PGY ;
 EV2= EVOL TURQ  MANU ABS PGX ORD PGT ;
 EV = EV1 ET EV2;
 SI(NEG GRAPH 'N');
  DESS EV;
 FINSI ;
*
TEMPS ;
*             CODE FONCTIONNEMENT
RESI1=(MAXI ABS (PGY - PGT ))/ VALK ;
SI(RESI1 <EG 5E-5);
MESS '    ';
MESS '    ';
MESS '    ';
MESS '    ';

  MESS ' COMPARAISON RESULTATS THEORIQUES ET CALCULES';
MESS '    ';
  MESS ' JUSQU AU TEMPS 4. : PRECISION < A 5E-5 PRES';
SINO;
    ERRE  5;
FINSI;
FIN;
```

3.9 Test plas10 Description sheet

Test plas10 **Description sheet**

Test name plas10

Calculation type MECHANICS ELASTIC PLASTIC - CAM-CLAY'S MATERIAL

Finite element type CUB8

Topic Tensile-test on a parallelipiped.

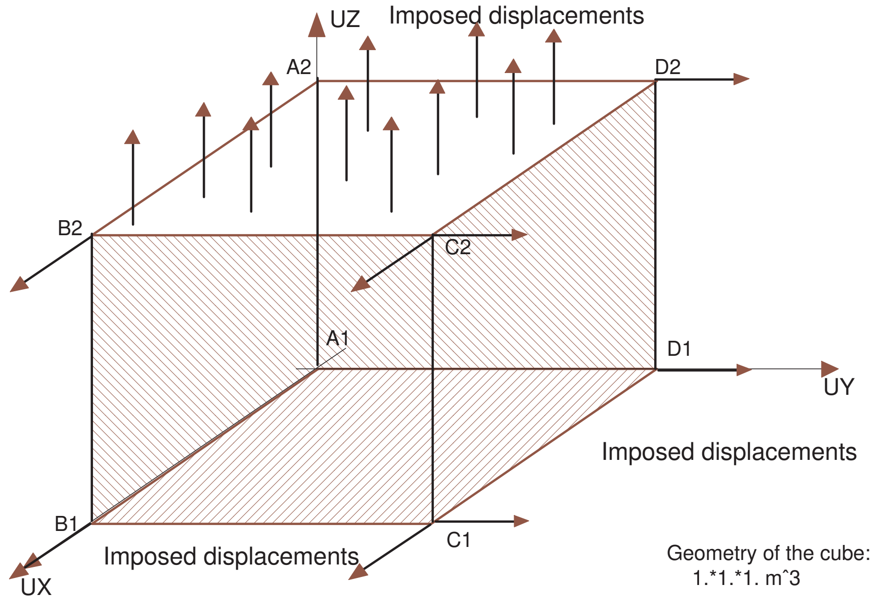
An isostatic embedded parallelipiped is subjected to imposed displacement on three of its faces. It follows the Cam-Clay model.

Goal Compare the hydrostatic compressive curve with the exact solution.

Version 97' customer version

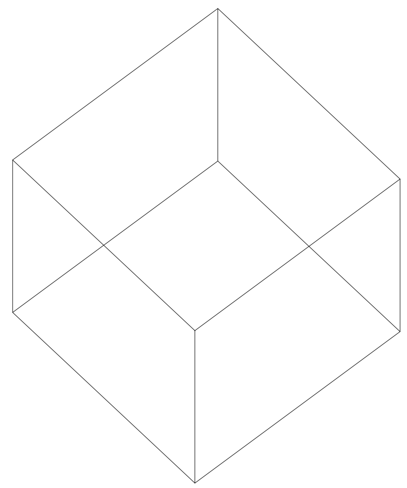
Model description

YOUNG	1000
NU	0.2

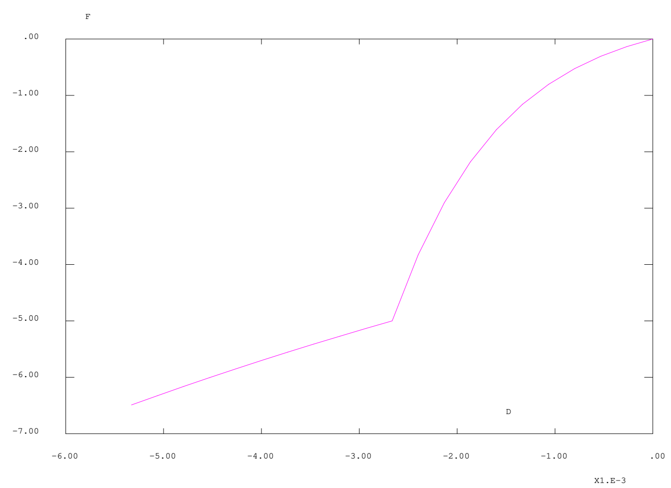


Test plas10 **Results**

CASTEM FIGURES



PARALLELEPIPEDE EN COMPRESSION HYDROSTATIQUE



PARALLELEPIPEDE EN COMPRESSION HYDROSTATIQUE

```
*          Test Plas10.dgibi: Jeux de données       *
*          --------------------------------         *
*                                                   *
OPTION ECHO 0;
GRAPH='N';
SAUT PAGE ;
*
*******************************************************
*  TEST PLAS10
*
* Sortie du domaine élastique et phase plastique
* (comportement élasto-plastique modèle CAM-CLAY).
*
* Un parallelépipède est soumis à des déplacements
* imposés sur trois de ses faces et fixé de manière
* isosattique. On compare la courbe de réponse en
* compression hydrostatique à la solution exacte.
*
*******************************************************

TITRE 'PARALLELEPIPEDE EN COMPRESSION HYDROSTATIQUE ' ;
OPTION ECHO 0 ;
OPTION DIME 3 ELEM CUB8 MODE TRID      ;
*-----------------------------------------------------
*          DEFINITION DE LA GEOMETRIE
*-----------------------------------------------------
A1= 0. 0. 0.;
B1= 1. 0. 0.;
C1= 1. 1. 0.;
D1= 0. 1. 0.;
A2= 0. 0. 1.;
B2= 1. 0. 1.;
C2= 1. 1. 1.;
D2= 0. 1. 1.;
*
L1 = A1 DROIT 1 B1  ;
L2 = B1  DROIT 1 C1  ;
L3 = C1  DROIT 1 D1  ;
L4 = D1  DROIT 1 A1  ;
LL = L1 ET L2 ET L3 ET L4 ;
SU1= SURF LL PLAN ;
M1 = A2 DROIT 1 B2  ;
M2 = B2  DROIT 1 C2  ;
M3 = C2  DROIT 1 D2  ;
M4 = D2  DROIT 1 A2  ;
MM = M1 ET M2 ET M3 ET M4 ;
SU2= SURF MM PLAN ;
SU = SU1 VOLU  1 SU2 ;
SI(NEG GRAPH 'N');
  TRAC (888  1000 2000 )  SU ;
FINSI ;
*-----------------------------------------------------
*       DEFINITION DES CONDITIONS AUX LIMITES
*            ET DES DEPLACEMENTS IMPOSES
*-----------------------------------------------------
BLX = BLOQ UX ( A1  ET A2 ET D1 ET D2 );
BLY = BLOQ UY ( A1 ET B1 ET B2 ET A2 ) ;
BLZ = BLOQ UZ SU1 ;
BL = BLX ET BLY ET BLZ  ;
*
CLX = BLOQ UX ( B1  ET B2 ET C1 ET C2 );
CLY = BLOQ UY ( D1 ET C1 ET C2 ET D2 ) ;
CLZ = BLOQ UZ SU2 ;
CL = CLX ET CLY ET CLZ  ;
*
VAL = -2.664328E-5 ;
DEPX= DEPI CLX  VAL   ;
DEPY= DEPI CLY  VAL   ;
DEPZ= DEPI CLZ  VAL   ;
*
*-----------------------------------------------------
*       DEFINITION DES PROPRIETES MATERIELLES
*-----------------------------------------------------
MO  = MODE SU MECANIQUE ELASTIQUE PLASTIQUE CAM_CLAY ;
*
XL = 0.1 ;
CC = 0.5 ;
XK = 0.01 ;
EE = 2. ;
PI0 = 5.;
*
MAT  = MATR MO  'YOUN' 1000.  'NU' 0.2
 'E0' EE  'COHE' CC  'M' 1.2   'P0' PI0   'LAMD' XL
 'KAPA' XK 'G1'  416.666667 ;
*
LITX = PROG 0. PAS 1. 300. ;
LIX =  PROG 0. PAS 1. 300. ;
EVX= EVOL MANU 'TEMP' LITX 'FX'   LIX ;
CHA1 = CHAR 'DIMP' (DEPX ET DEPY ET DEPZ) EVX;
*
*-----------------------------------------
*          APPEL A PASAPAS
* AU TEMPS T , LE DEPLACEMENT VAUDRA F(T)*D
*-----------------------------------------
 TAB1 = TABLE ;
 LIS1 = PROG 0. PAS 10.  200. ;
 TAB1.'MODELE' = MO;
 TAB1.'CARACTERISTIQUES' = MAT;
 TAB1.'CHARGEMENT' = CHA1;
 TAB1.'TEMPS_CALCULES' = LIS1;
 TAB1.'BLOCAGES_MECANIQUES' = CL 'ET' BL;
 PASAPAS TAB1 ;
*
*-----------------------------------------
*          DEPOUILLEMENT ET
*    CALCUL DE LA SOLUTION ANALYTIQUE
*-----------------------------------------
*
SI(NEG GRAPH 'N');
NT = (DIME (TAB1.DEPLACEMENTS)) - 1 ;
IT = 0 ;
PRD = PROG 0. ;
PRF = PROG 0. ;
REPETER BOPI NT;
IT = IT + 1 ;
DD= TAB1.DEPLACEMENTS.IT;
RR =  REAC CLY DD;
RRR = RESU RR;
LEP = ( EXTR RRR 'MAIL' 'NOMU'  ) POIN 1  ;
PRD = PRD ET (PROG ( EXTR DD UY D1));
PRF = PRF ET (PROG ( EXTR RRR FY LEP) ) ;
FIN BOPI ;
FV = EVOL ROSE MANU 'D' PRD 'F ' PRF;
DESS FV ;
FINSI;
*
VV = PECHE TAB1 VARIABLES_INTERNES 200. ;
SS = PECHE TAB1 CONTRAINTES 200. ;
PCAL =  EXTR SS SMXX 1 1 1  ;
PTHE =  -1. * ( PI0 + CC) *
       (EXP ( ( -300. * VAL ) * ( 1.+EE ) / XL) );
PTHE = PTHE + CC ;
EVPCAL =  EXTR VV TRAP 1 1 1  ;
EVPTHE = (300. * VAL ) * ( XL - XK ) / XL ;
SAUTER 3 LIGNE ;
MESS ' PRESSION MOYENNE THEORIQUE ' PTHE
```

```
'  CALCULEE'  PCAL;
MESS ' TRACE PLASTIQUE THEORIQUE ' EVPTHE
' CALCULEE ' EVPCAL;
*
RESI1 = ABS ( (PCAL-PTHE)/PTHE );
RESI2 = ABS ( (EVPCAL-EVPTHE)/EVPTHE);
SAUTER 3 LIGNE ;
MESS ' ERREURS RELATIVES ' RESI1 RESI2 ;
*
*              CODE FONCTIONNEMENT
*
SI (RESI1 <EG 1.E-6) ;
  SI (RESI2 <EG 1.E-6) ;
    ERRE  0;
  SINO;
    ERRE  5;
  FINSI;
SINON;
    ERRE  5;
FINSI;
FIN;
```

3.10 Test plas11 Description sheet

Test plas11 **Description sheet**

Test name plas11

Calculation type MECHANIC ELASTIC PLASTIC - GENERALISED PLANE STRAINS

Finite element type QUA8, COQ2, POI1 **Topic** Beam subjected to a rotation.

A rectangular cross-section beam is subjected to an imposed rotation RX. The beam is modelled with a massive piece (1x1), with a COQ2 element piece (length: .5 thickness: 2.) and with a piece made of POI1 point elements (4 points, 2 in the upper piece and 2 in the lower one's).

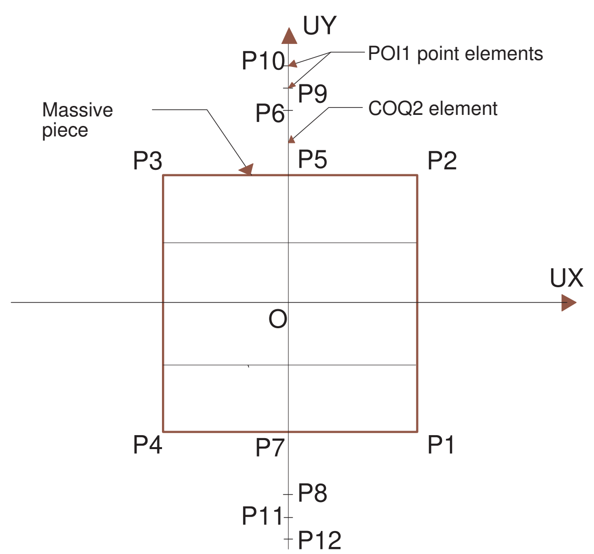
Goal Study the behavior of the resulting moment of this imposed rotation up to the yield point.

Version 97' customer version

Model description

	Massive piece	COQ2 elements	POI1 point elements
YOUNG	2. 10^{11} Pa	2. 10^{11} Pa	2. 10^{11} Pa
NU	0.	0.	0.
SIGY	4.83 10^8 Pa	4.83 10^8	4.83 10^8

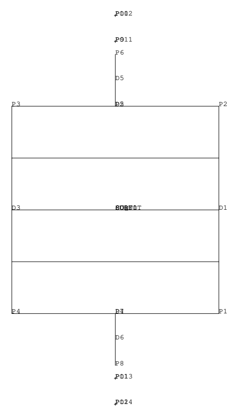
Imposed rotation



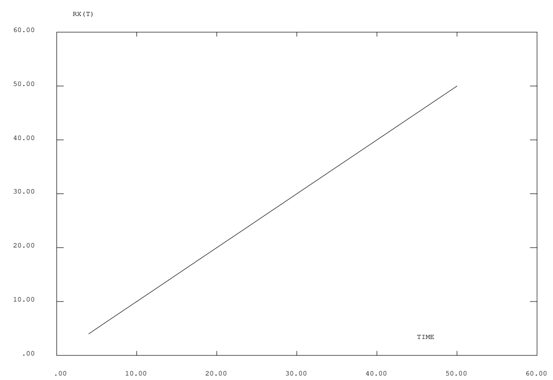
Geometry of the rectangular cross-section beam
thickness: 2.
heigth: 4.

Test plas11 **Results**

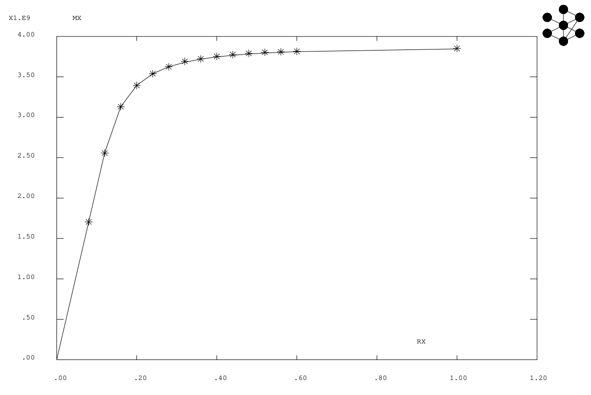
CASTEM FIGURES



PLAS12 : MESHING



PLAS11: LOAD



PLAS11 : COURBE MX EN FONCTION DE RX

```
*          Test Plas11.dgibi: Jeux de données
*          ---------------------------------
*
OPTION ECHO 1;
GRAPH = 'N';
SAUT PAGE;
*
*******************************************************
*                                                     *
*      TEST : PLAS11                                  *
*      DEFORMATIONS PLANES GENERALISEES               *
*                                                     *
*    une poutre de section rectangulaire est          *
*    soumise à une rotation imposée RX :              *
*                                                     *
*     Hauteur:4 Largeur:2                             *
*                                                     *
*    ,,,,,,,,,                                        *
*    '   *   '                                        *
*    '   |   '           cette poutre est modélisée   *
*    ---------          avec une partie massive (1x1) *
*    |       |          une autre avec des éléments   *
*    |   -------->@RX  COQ2  (long :.5 epais 2.) et   *
*    |       |          la dernière avec des éléments *
*    ---------          point POI1 : 4 point, 2 en    *
*    '   |   '          haut, 2 en bas (sect .5 )     *
*    '   *   '                                        *
*    ,,,,,,,,,                                        *
*                                                     *
*    On étudie le comportement du moment résultant    *
*    de cette rotation imposée jusqu'au domaine       *
*    plastique                                        *
*                                                     *
*******************************************************
TITRE 'PLAS11 - SECTION DE POUTRE SOUMISE A UNE
ROTATION IMPOSEE';
OPTION ECHO 1;
TEMPS;
*
*
OPTI DIME 2;
OPTI ELEM QUA8;
*
* on définit le point support des déformations
* planes généralisées
*
PG = 0. 0.;PG1=PG PLUS (0. 0.);PG2 = PG PLUS (0. 0.);
*
OPTI MODE PLAN GENE  ;
*
* quelques initialisations :
*
I = 1;
EVL1 = PROG 0. ;
EVL2 = PROG 0. ;
EVL3 = PROG 0. ;
*
* et quelques constantes pour le calcul théorique :
*
* demi hauteur de la section
H = 2.;
* demi largeur de la section
B = 1. ;
* le module de young
E1 = 2E11;
* la limite élastique du matériau
SILIM = 4.83E8;
* le moment d'inertie par rapport a l'axe des X
```

```
* IX = (2.*B)*((2.*H)**3)/12.;
* * la valeur limite de la déformation élastique
* EPSEL = SILIM/E1;
  * la valeur limite du moment élastique
  MOMEL = E1*IX*EPSEL/H;
  * le moment plastique ultime
  MOLIM = 3*MOMEL/2.;
  *
* *=====================================================
* * géometrie : maillage
* *=====================================================
* *
* * dimension en mètres
* *
* *----------maillage partie massive-----------*
* *
* * points
* *
* P1 = 1. -1.;
* P2 = 1. 1.;
* P3 = -1. 1.;
* P4 = -1. -1.;
* *
* * droite et lignes
* *
* * coefficients de finesse du maillage
* *
* N = 4;
* M = 1;
* *
* D1 = P1 D N P2 ;
* D2 = P2 D M P3 ;
* D3 = P3 D N P4 ;
* D4 = P4 D M P1 ;
  *
  SURF1 = DALL D1 D2 D3 D4;
  *
  *----------maillage éléments coq2------------*
  *
  OPTI ELEM SEG2 ;
  *
  P5 = 0. 1. ;
  P6 = 0. 1.5 ;
  P7 = 0. -1. ;
  P8 = 0. -1.5 ;
  *
  D5 = P5 D 2 P6;
  D6 = P7 D 2 P8;
  COQU1 = D5 ET D6 ;
  *
  P9 = 0. 1.625;
  P10 = 0. 1.875;
  P11 = 0. -1.625;
  P12 = 0. -1.875;
  *
  *----------maillage éléments poi1------------*
  *
  PO11 = MANU POI1 P9;
  PO12 = MANU POI1 P10;
  PO13 = MANU POI1 P11;
  PO14 = MANU POI1 P12;
  PO1 = PO11 ET PO12 ET PO13 ET PO14;
  *
  SURTOT = SURF1 ET COQU1 ET PO1 ;
  *
  SI (NEG GRAPH 'N');
     TITR 'PLAS12 : MAILLAGE';
     TRAC QUAL SURTOT;
```

```
FINSI;
*
ELTOT = NBEL SURTOT;
NOTOT = NBNO SURTOT;
*
*
*=======================================================
* modèle - matériau
* rigidité
*=======================================================
*
*----------------partie massive-----------------*
*
MO1 = MODE SURF1 MECANIQUE ELASTIQUE PLASTIQUE PARFAIT
      QUA8 DPGE PG;
MA1 = MATE MO1 YOUN E1 NU 0. SIGY SILIM;
*
*----------------éléments coq2------------------*
*
MO2 = MODE COQU1 MECANIQUE ELASTIQUE PLASTIQUE PARFAIT
      COQ2 DPGE PG1;
MAT2 = MATE MO2 YOUN E1 NU 0. SIGY SILIM;
CAR2 = CARA MO2 EPAI 2. ;
MA2 = MAT2 ET CAR2 ;
*
*-----------------éléments poi1-----------------*
*
MO3 = MODE PO1 MECANIQUE ELASTIQUE PLASTIQUE PARFAIT
      POI1   DPGE PG1;
MAT3 = MATE MO3 YOUN E1 NU 0. SIGY SILIM;
CAR3 = CARA MO3 SECT .5 ;
MA3 = MAT3 ET CAR3;
*
MODE1 = MO1 ET MO2 ET MO3;
MATE1 = MA1 ET MA2 ET MA3;
*
RI1 = RIGI MODE1 MATE1;

*
*=======================================================
* conditions aux limites
*=======================================================
*
P15 = SURF1 POINT PROCHE (0. 0.);
P16 = SURF1 POINT PROCHE (0. 1.);
B1 = BLOQ DEPL P15;
B2 = BLOQ UX P16;
B3 = BLOQ DEPL PO1 ;
B4 = BLOQ UX COQU1 ;
B5 = BLOQ UY (P5 ET P7) ;
B6 = BLOQ UZ RY (PG et pg1 );
B7 = BLOQ RX ( PG et pg1 );
BTOT = B1 ET B2 ET B3 ET B4 ET B5 ET B6 ET B7;
RI = RI1 ET BTOT;
*
*=======================================================
* chargement
*=======================================================
*
TEMP1 = (PROG 4. PAS 2. 30.) ET (PROG 50.);
LAM1 = (PROG 4. PAS 2. 30.) ET (PROG 50.);
FAT1 = .0002;
TITRE 'PLAS11 : CHARGEMENT';
EV1 = EVOL MANU 'TEMPS' TEMP1 'RX(T)' LAM1 ;
DE1 = DEPI B7 FAT1 ;
CHRIX = CHAR 'DIMP' DE1 EV1 ;
*
SI (NEG GRAPH 'N');
  DESS EV1 ;
FINSI;
*
*=======================================================
* calcul non linéaire
*=======================================================
*
TAB1 = TABLE;
TAB1.'BLOCAGES_MECANIQUES' = BTOT;
TAB1.'MODELE' = MODE1;
TAB1.'CARACTERISTIQUES' = MATE1;
TAB1.'CHARGEMENT' = CHRIX;
TAB1.'TEMPS_CALCULES' = TEMP1;
tab1.'PRECISION' = 1.e-6;

PASAPAS TAB1 ;
*
*=======================================================
* post-traitement : tracé d'une courbe et comparaison
* avec la solution analytique
*=======================================================
*
DIM1 = DIMENSION TEMP1;
*
REPETER BLOC1 DIM1;
*
TPS = EXTR TEMP1 I;
CH1 = PECHE TAB1 DEPLACEMENTS TPS;
MOIMP = REAC RI CH1;
*
*EXTRACTION DU MOMENT MX CORRESPONDANT À LA ROTATION RX
*
MMX = EXTR MOIMP MX PG;MMX1= EXTR MOIMP MX PG1;MMX=MMX+MMX1;
RRX = EXTR CH1 RX PG;RRX1= EXTR CH1 RX PG1;RRX=RRX+RRX1;
EVL1 = EVL1 ET (PROG MMX );
EVL2 = EVL2 ET (PROG RRX );
*
*---------------SOLUTION THEORIQUE--------------------*
*
EPSN = 2.*RRX ;
*
* quand on est dans le domaine plastique :
*
SI (EPSN > EPSEL) ;
 MOMN = (1 - ((( EPSEL / EPSN) ** 2) / 3)) * MOLIM ;
 EVL3 = EVL3 ET (PROG MOMN);
*
* quand on est dans le domaine élastique :
*
SINON;
 MOMN = E1 * IX * RRX ;
 EVL3 = EVL3 ET (PROG MOMN) ;
FINSI;
*
I = I+1;
*
FIN BLOC1;
*
SI (NEG GRAPH 'N');
*
COURB = TABLE;
COURB.1 = 'MARQ ETOI NOLI' ;
*
titre 'PLAS11 : COURBE MX EN FONCTION DE RX ';
EV1 = EVOL MANU 'RX' EVL2 'MX' EVL1 ;
EV2 = EVOL MANU 'RX' EVL2 'MX' EVL3 ;
*  courbe théorique : traits continus
* point calculés    : marques étoiles
```

```
DESS  (EV1 ET EV2) COURB LOGO;
FINSI;
*
* calcul de l'écart avec la solution de référence
*
ERG1 = 100*(ABS((MOMN-MMX)/MOMN));
*
*=======================================================
* affichage des résultats
*=======================================================
*
MESS ' RESULTATS : TEST PLAS11 ';
MESS '--------------------------';
SAUT 2 LIGN;
*
MESS 'on cherche la valeur du moment calculé dans le ';
MESS 'domaine plastique';
MESS 'elle est comparée a une valeur obtenue
analytiquement';
SAUT 1 LIGN;
MESS ' moment théorique :' MOMN 'MPa';
SAUT 1 LIGN;
MESS ' moment calculé :' MMX 'MPa';
SAUT 1 LIGN;
MESS '          Soit un écart de :' ERG1 '%';
SAUT 2 LIGN;
*
MESS '    Nombre d elements :' ELTOT;
MESS '    Nombre de noeuds  :' NOTOT;
MESS '       soit ' (NOTOT * 3) 'd.d.l.';
*
*
*=======================================================
*                  code de fonctionnement
*=======================================================
*
*     L'écart maximum entre valeur théorique et
* calculée doit être inférieur a 0.5%.
*
SAUT 2 LIGN;
SI (ERG1 <EG 0.5);
    ERRE 0;
SINON;
    ERRE 5;
FINSI;
*
*=======================================================
*                 Temps de calcul et fin
*=======================================================
*
SAUT 1 LIGN;
TEMPS;
SAUT 1 LIGN;
*
FIN;
*
```

3.11 Test plas12 Description sheet

Test plas12 **Description sheet**

Test name plas12

Calculation type MECHANICS ELASTIC PLASTIC

Finite element type TUYA

Topic Pipe subjected to thermal shock.

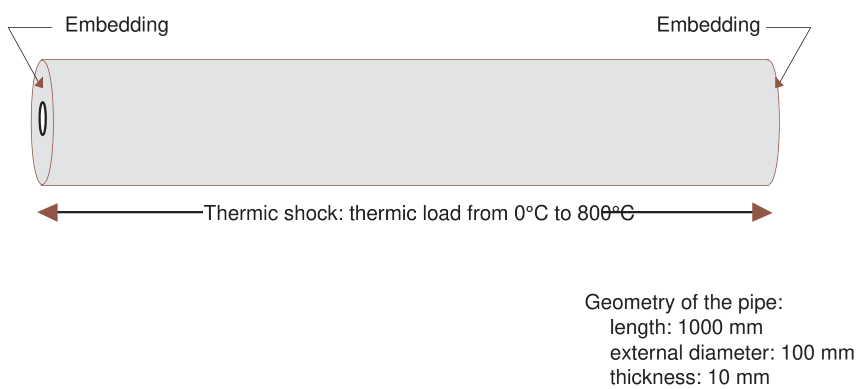
A pipe is embedded at its two ends and subjected to a thermic shock.

Goal Test the PASAPAS procedure with a material depending on the temperature. Compare the Castem's result for the stress with the analytical solution:
EPSILLON = Alpha * (Delta T)
SIGMA = Interpolation of the EPSILLON value on the tensile curve

Version 97' customer version

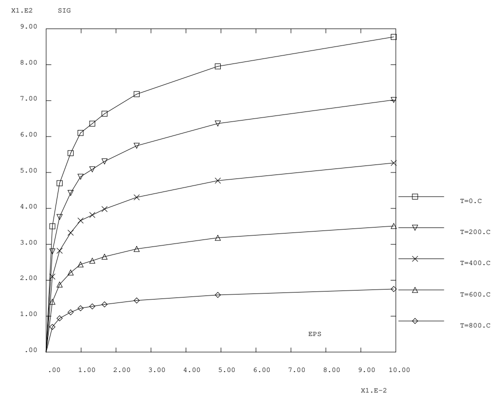
Model description

Material properties	
YOUNG	function of the temperature
NU	0.3
Thermic expansion coefficient	function of the temperature
Tensile curve	function of the temperature

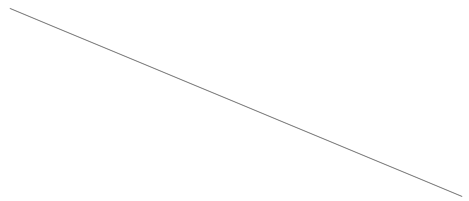


Test plas12
Results

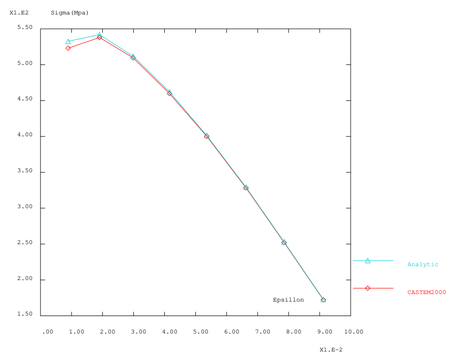
CASTEM FIGURES



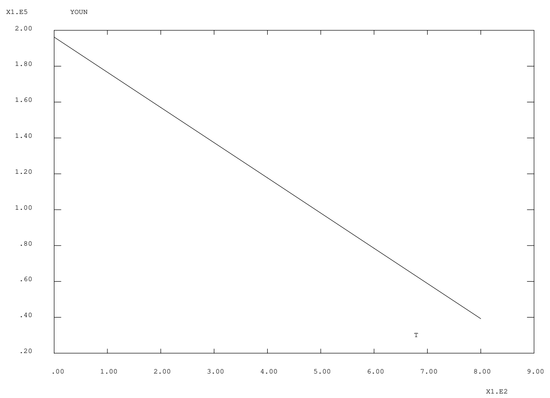
Tensile curve as a function of temperature



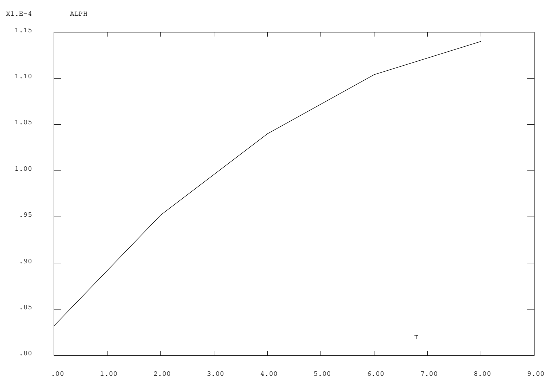
Themic expannsion as a function of the temperature



Themic expannsion as a function of the temperature



Modulus of Young as a function of the temperature



Themic expannsion as a function of the temperature

```
*          Test Plas12.dgibi: Jeux de données       *
*          --------------------------------         *
*                                                    *
GRAPH='N';
SAUT PAGE ;
*
*******************************************************
* TEST PLAS12
*
* Un tuyauderie encastré en deux extremités
* soumis à un choc thermique.
*
* GEOMETRIE : Longueur du tuyau l         : 1000. mm
*            Rayon extérieure du tuyau r : 10.   mm
*            Epaisseur du tuyau e        : 100.  mm
* MATERIAU  : Module d'Young E            : dépendant
*                                          de la
*                                          température
*            Coeficient de Poisson Nu    : 0.3
*            dilatation thermique Alpha  : dépendant
*                                          de la
*                                          température
*            Courbe de traction          : dépendant
*                                          de la
*                                          température
* CHARGEMENT: Une tempérarure T uniforme passant
*            de 0C à 800C
*
*    encastrement                         encastrement
*        |-------------------------------------|
*        p1        == CHOC THERMIQUE ==        p2
*
*******************************************************
* Objectif du test:
* Verifier la procédure PASAPAS avec un matériau
* dépendant de la température.
*******************************************************
* Validation du calcul:
* On compare la contrainte de PASAPAS avec la solution
* exacte : EPSILLON = Alpha * (Delta T)
*         SIGMA = Interpolation de la valeur EPSILLON
*                 sur la courbe de traction
*******************************************************
*******************************************************
* procédure calculant le module D'Young avec une courbe
* de traction
     DEBP CALYO EV1*EVOLUTION;
       X1=EXTR EV1 ABSC;
       Y1=EXTR EV1 ORDO;
       YOU1=(EXTR Y1 2)/(EXTR X1 2);
     FINP YOU1;
*******************************************************
*
* 1) option de calcul
     opti dime 3 elem seg2;
*
* 2) données géométriques
     l=1000.; e = 10; r = 100.;
*
* 3) données de caractéristiques du matériau
**** 3.1) courbe de traction en fonction de la
****      température
     TITR 'Courbe de traction en fonction de la
température';
     progx = prog 0. 0.001785 0.00386 0.00701 0.00984
             0.0132 0.0167 0.0259 0.0491 0.0994;
     progy = prog 0. 350.23 470.05 553.73 610.14
             635.94 663.59 717.97 795.39 877.42;
     ev1 = evol manu eps progx sig (progy*1.0);
     ev2 = evol manu eps progx sig (progy*0.8);
     ev3 = evol manu eps progx sig (progy*0.6);
     ev4 = evol manu eps progx sig (progy*0.4);
     ev5 = evol manu eps progx sig (progy*0.2);
     tra = nuage comp T    0.  200. 400. 600. 800.
                 comp TRAC ev1 ev2  ev3  ev4  ev5;
     SI (NEG GRAPH 'N');
       TAB1=TABL;
       TAB1.1 = 'MARQ LOSA ';
       TAB1.2 = 'MARQ TRIB ';
       TAB1.3 = 'MARQ CROI ';
       TAB1.4 = 'MARQ TRIA ';
       TAB1.5 = 'MARQ CARR ';
       TAB1.'TITRE'=TABL;
       TAB1.'TITRE'. 1 = MOT 'T=800.C';
       TAB1.'TITRE'. 2 = MOT 'T=600.C';
       TAB1.'TITRE'. 3 = MOT 'T=400.C';
       TAB1.'TITRE'. 4 = MOT 'T=200.C';
       TAB1.'TITRE'. 5 = MOT 'T=0.C';
       DESS (ev5 et ev4 et ev3 et ev2 et ev1) lege tab1;
     FINSI;
**** 3.2) Module D'Young en fonction de la température
     TITR 'Module d Young en fonction de la
température';
     y1=calyo ev1;
     y2=calyo ev2;
     y3=calyo ev3;
     y4=calyo ev4;
     y5=calyo ev5;
     evy = evol manu T   (prog 0.  200. 400. 600. 800.)
                    YOUN (prog y1  y2   y3   y4   y5);
     SI (NEG GRAPH 'N'); DESS evy; FINSI;
**** 3.3) Alpha en fonction de la température
     TITR 'Dilatation thermique en fonction de la
température';
     eva = evol manu T  (prog 0.   200. 400. 600. 800.)
          ALPH ((prog 10.4 11.9 13.0 13.8 14.25)*8E-6);
     SI (NEG GRAPH 'N'); DESS eva; FINSI;
**** 3.4) la section du tuyau
     s = pi*((r*r) - ((r - e)*(r - e)));
* 4) maillage
     TITR 'Tuyauderie encastré soumis à un choc
thermique';
     p1 = 0 0 0; p2 = L 0 0;
     d1= p1 d 10 p2;
     SI (NEG GRAPH 'N'); TRAC D1; FINSI;
*
* 5) Température de référence
     TREFE = MANU CHPO d1 1 T 0.;
*
* 6) modele, materiaux et caracteristiques geometriques
     modti=modl d1 mecanique elastique plastique
           isotrope tuya;
     mater=matr modti 'YOUN' evy 'NU' 0.3 'ALPH' eva
           'TRAC' tra;
     carac=carb modti 'EPAI' e 'RAYO' r 'CISA' 1. ;
     mc=mater et carac;
*
* 6) bloquage
     bloqtot=bloq depl rota (p1 et p2) ;
*
* 8) chargement thermique
     TBTEMPE = TABLE;
     TBTEMPE.0 =TREFE;
     TBTEMPE.1 =MANU CHPO d1 1 T 200.;
     TBTEMPE.2 =MANU CHPO d1 1 T 400.;
     TBTEMPE.3 =MANU CHPO d1 1 T 600.;
```

```
    TBTEMPE.4 =MANU CHPO d1 1 T 800.;
    TEMP = TABLE;
    TEMP.0 = 0.;
    TEMP.1 = 2.;
    TEMP.2 = 4.;
    TEMP.3 = 6.;
    TEMP.4 = 8.;
    EVT = EVOL MANU TEMPS (prog 0. 2.   4.   6.   8.)
                    T   (prog 0. 200. 400. 600. 800.);
    CH2 = CHAR T TEMP TBTEMPE;
*
* 9) chargement mecanique
    f1 = forc (0. 0. 0.) p1;
    ch1 = char meca f1 (evol manu (prog 0. 10.)
         (prog 0. 0.));
    chat = ch2 et ch1;
*
* 10) calcul nonlinéaire
    TAB1 = TABLE;
    TAB1.'TEMPERATURE_REFERENCE'=TREFE;
    TAB1.'MODELE' = modti;
    TAB1.'CARACTERISTIQUES' = MC;
    TAB1.'BLOCAGES_MECANIQUES' = BLOQTOT;
    TAB1.'TEMPS_CALCULES' = (prog 0. pas 1. 8.);
    TAB1.'CHARGEMENT' = CHAT;
    PASAPAS TAB1;
*
* 11) comparaison et message
    TBEVOL=TABL;
    TBEVOL.1=(EV1 + EV2)/2.;
    TBEVOL.2=EV2;
    TBEVOL.3=(EV2 + EV3)/2.;
    TBEVOL.4=EV3;
    TBEVOL.5=(EV3 + EV4)/2.;
    TBEVOL.6=EV4;
    TBEVOL.7=(EV4 + EV5)/2.;
    TBEVOL.8=EV5;
    TBCONT=TAB1.CONTRAINTES;
    PCALCUL=PROG; PANALYT=PROG; PEPSI=PROG;
    REPETER B1 ((DIME TBCONT) - 1);
      V1 = (EXTR (TBCONT.(&B1)) EFFX 1 1 1)/S;
      PCALCUL=PCALCUL ET (PROG (ABS V1));
      T1 = IPOL (EXTR EVT ABSC) (EXTR EVT ORDO)
           (TAB1.TEMPS.&B1);
      Y1 = IPOL (EXTR EVY ABSC) (EXTR EVY ORDO) T1;
      A1 = IPOL (EXTR EVA ABSC) (EXTR EVA ORDO) T1;
      PEPSI=PEPSI ET (PROG (A1*T1));
      EVC=TBEVOL.&B1;
      V2 = IPOL (EXTR EVC ABSC) (EXTR EVC ORDO) (A1*T1);
      PANALYT=PANALYT ET (PROG V2);
    FIN B1;
    ECALCUL=EVOL ROUG MANU 'Epsillon' PEPSI
            'Sigma(Mpa)' PCALCUL;
    EANALYT=EVOL TURQ MANU 'Epsillon' PEPSI
            'Sigma(Mpa)' PANALYT;
    TAB1 = TABL;
    TAB1.1 = 'MARQ LOSA ';
    TAB1.2 = 'MARQ TRIB ';
    TAB1.'TITRE' = TABL;
    TAB1.'TITRE'. 1 = MOT 'CASTEM2000';
    TAB1.'TITRE'. 2 = MOT 'Analytique';
    SI (NEG GRAPH 'N');
      DESS (ECALCUL et EANALYT) lege tab1;
    FINSI;
    ENEANA=EXTR 1 (SOMM EANALYT);
    ENEK20=EXTR 1 (SOMM ECALCUL);
    ERR1=ABS ((ENEANA - ENEK20)/ENEANA);
    mess '***********************';
mess 'Energie de déformation cumulée';
mess 'k2000,theorie,erreur:  ' ENEK20 ENEANA ERR1;
mess '***********************';
si (ERR1 > 5.5d-3);
  erre 5;
sinon;
  erre 0;
finsi;
fin;
```

3.12 Test plas13 Description sheet

Test plas13 **Description sheet**

Test name plas13

Calculation type MECHANICS ELASTIC PLASTIC

Finite element type POUT

Topic Material of Ambrois's post type

A post is subjected to a 2-tun load. The material is an Ambrois's post type. It is a global plasticity model for reinforced concrete beams which makes it possible to take into account non linear behavior laws according to the type of loading (axial, bending and shear).

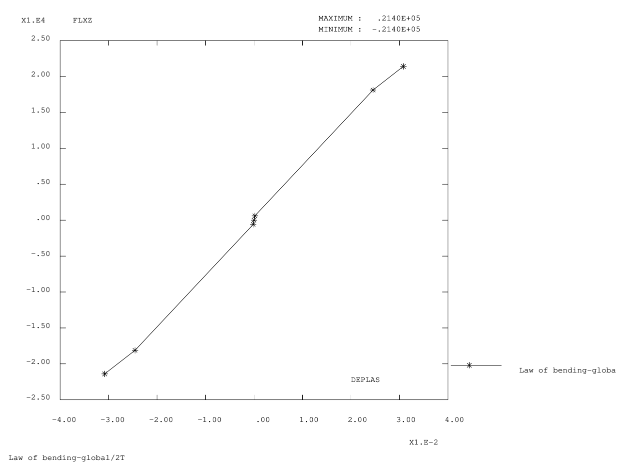
Goal Study an Ambroy's material with GLOBAL OLARIU elements.

Version 97' customer version

Model description

Material properties	
YOUNG	$35510\ 10^6$
NU	0.2
RHO	2516.

CASTEM FIGURES



```
*          Test Plas13.dgibi: Jeux de données       *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0;
OPTI DIME 3 ELEM SEG2 MODE TRID;
*******************************************************
*                                                     *
* COMPARAISON ETUDE AMBROIS AVEC ELEMENT GLOBAL OLARIU*
*                                                     *
*******************************************************
*      DEFINITION VALEURS ET ELEMENTS GRAPHIQUES      *
*******************************************************
*
* MATERIAU DU TYPE  POTEAU AMBROIS
**********************************
*
*POTEAU SOUS 2 T
*******************
*
SIGY = PROG  -21.410E6  -25.208E6  -22.904E6
-1.4202E6  0. 1.4202E6   22.904E6   25.208E6 21.410E6 ;
EPSY = PROG  -1.80E-2  -8.50E-3  -3.0E-3  -0.40E-4  0.
                0.40E-4   3.0E-3   8.50E-3   1.80E-2  ;
*
TITRE 'MATERIAU AMBROIS POTEAU SOUS 2 T' ;
EVT = EVOL ROUGE MANU 'EPS' EPSY  'SIG' SIGY  ;
  ;
*******************************************************
*                         M A I L L A G E 1         *
*******************************************************
*
DENS 1 ;
*
A0 =  0.  0.  0.    ;
B0 =  0.  0.  1.50  ;
*
*******************************************************
*              DESCRIPTION EN MODELE GLOBAL         *
*******************************************************
*
****************
* LES POTEAUX  *
****************
*
POT = A0 D 1 B0 ;
*
MOPOTA = MODL POT MECANIQUE ELASTIQUE PLASTIQUE GLOBAL
         POUTRE ;
*
*POTEAU PRECONTRAINT SOUS 2 TONNES
**********************************
MOM = PROG -21.401E3 -18.120E3  -0.5961E3 0.
        0.5961E3 18.120E3 21.401E3 ;
DPLAS = PROG -3.08E-2 -2.455E-2 -1.764E-4 0.
         1.764E-4 2.455E-2  3.08E-2 ;
*
TITRE 'LOI-FLEXION-GLOBAL/2T';
TC=TABLE;
EVE = EVOL BLAN MANU 'DEPLAS' DPLAS 'FLXZ' MOM;
*TC.1 = 'MARQ ETOI';
*DESS EVE CARR LEGE MIMA TC;
*
*MATERIAU DE TYPE GLOBAL
*************************
*
MAPOTA = MATR MOPOTA 'SECT'0.0289
  'INRY' 7.137E-5 'INRZ' 7.137E-5
  'TORS' 1.178E-4 'SECY' 0. 'SECZ' 0.
   'DY' 0.085  'DZ' 0.085
  'YOUNG' 35510.E6 'NU' 0.2 'RHO' 2516.
   FLXZ EVE 'VECT' ( 0. -1. 0. );
*
*CONDITIONS AUX LIMITES
**********************
*
RIG = RIGI MOPOTA MAPOTA ;
*
BL11 = BLOQ DEPL ROTA ( A0 );
BL12 = BLOQ UX UZ RY RZ ( B0 );
BL1 = BL11 ET BL12 ;
*
RIGV = RIG ET BL1  ;
*
MAS  = MASSE MOPOTA MAPOTA ;
MAS1 = MASSE UY  2.0E3 (B0);
MAS= MAS ET MAS1;
*
*  CALCUL DES  PREMIERS MODES
*******************************
*
*PREMOD1 = VIBRE 'PROC' (PROG 0.)  RIGV MAS TBAS;
PREMOD2 = VIBRE 'SIMULTANE' 0. 2  RIGV2 MASTOT2 TBAS;
*LIST PREMOD1.MODES . 1 ;
*
*  INTRODUCTION DU POIDS
************************
*
 V1    = MANU CHPO  POT 1  UZ -9.81 'NATURE' 'DIFFUS';
 FO1 = MAS * V1 ;
 VALTEM1 = PROG  0. 1.E-2 ;
 VALPOI  = PROG 1. 1. ;
 EV1 = EVOL  MANU 'TEMPS' VALTEM1 'POIDS' VALPOI  ;
 CHA1 = CHAR MECA FO1 EV1 ;
*
* INTRODUCTION DE L'ACTION DE CHARGEMENT STATIQUE POUR
******************************************************
* OBTENIR LA LOI DE COMPORTEMENT DE L'ELEMENT GLOBAL
******************************************************
*
 V2  = FORCE FY 1. B0 ;
AMPLIF1 = PROG 0. PAS 0.05 0.1;
AMPLIF2 = PROG 0.1 PAS 0.5 1.4266;
AMPLIF = AMPLIF1 ET AMPLIF2;
*
AMPLIF =  AMPLIF * 1.E4 ;
Q = DIME AMPLIF ;
VALTEM2 = PROG  1.E-2  PAS 1.E-2 (Q*1.E-2) ;
EV2 = EVOL  MANU 'TEMPS' VALTEM2 'CHARGEMENT' AMPLIF  ;
CHA2 = CHAR MECA V2 EV2 ;
*
* CALCUL STATIQUE AVEC POIDS UNIQUEMENT
* ************************************
*
 TAB = TABLE ;
 TAB.'CARACTERISTIQUES' = MAPOTA;
 TAB.'MODELE' = MOPOTA;
 TAB.'BLOCAGES_MECANIQUES' = BL1;
 TAB.'CHARGEMENT' = CHA1;
 TAB.'TEMPS_CALCULES' = VALTEM1;
 PASAPAS TAB ;
*
* ON ENCHAINE AVEC LE CHARGEMENT
* ******************************
*
 VALPOI  = PROG  Q  * 1. ;
 EV3 = EVOL  MANU 'TEMPS' VALTEM2 'POIDS' VALPOI  ;
 CHA3 = CHAR MECA FO1 EV3 ;
```

```
 CHA = CHA3 ET CHA2 ;
*
 TAB . 'DYNAMIQUE' = FAUX ;
 TAB.FTOL = 1.E-3;
 TAB.MTOL = 1.E-3;
 TAB.PRECISION = 1.E-5;
 TAB . 'MASSE' = MAS ;
 TPCAL = 1.E-2 ;
 TEMPCAL =  PROG 0. PAS TPCAL ((Q-1)*1.E-2);
 TAB.'CHARGEMENT' = CHA;
 TAB.'TEMPS_CALCULES' = VALTEM2;
 PASAPAS TAB ;
*
******************************
* RECUPERATION DES RESULTATS *
******************************
  tabd = TAB.DEPLACEMENTS ;
  tabc = TAB.CONTRAINTES ;
*
* No DU PAS
************
time = TAB.TEMPS;
ni = (DIME time) - 1;
*
RE = REAC BL1 (tabd.ni);
MOMAX = EXTR RE MX A0;
TEXT 'LE MOMENT MAX ATTEINT EST' MOMAX  'N.m' ;
*
*
PRECIS = 1.E-3;
BOOL1 = (MOMAX <EG ((1 + PRECIS)*21.401E3));
BOOL2 = (((1 - PRECIS)*21.401E3) <EG MOMAX );
BOOL =  BOOL1 ET BOOL2;
M1 = CHAINE 'PROBLEME DANS L_ELEMENT GLOBAL';
M2 = CHAINE ' TEST OK';
SI (BOOL EGA FAUX ) ;
MESS M1 ; ERRE 5 ;
SINON;
MESS M2;  ERRE 0 ;
FINSI;
*
FIN ;
```

3.13 Test plas14 Description sheet

Test plas14 **Description sheet**

Test name plas14

Calculation type MECHANICS ELASTIC PLASTIC ACIER_UNI AND BETON_UNI
MECHANICS ELASTIC FIBRE_NL FERAILLE AND FIBRE_NL BETON

Finite element type QUAS, TRIS

Topic Example of a concrete section with four 20-mm steel bars.

The ACIER_UNI (steel) model is a modified Menegotto-Pinto uni-axial model, which accounts for the steel buckling.
The BETON_UNI (concrete) model is a Hognestad-like model, accounting for the confinment.
The geometry of the concrete is as follows:

Concrete dimensions:
All section (0.25*0.25) m^2
Core section (0.20*0.20) m^2
Transversal reinforcement:
10 mm Bars / 7.0 cm

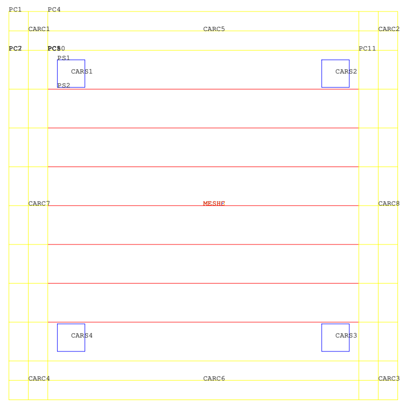
Version 97' customer version

Model description

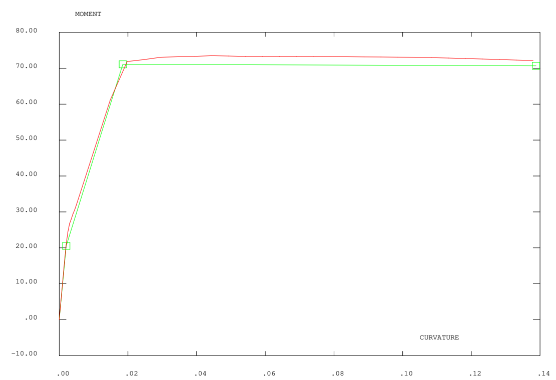
ACIER_UNI (Steel) model		
Young's modulus		2.03 10^5
Poisson's coefficient		0.30
Yielding stress		440.0
Ultimate strain		.090
Ultimate stress		760.0
Hardening strain		0.030
Ratio between the length of the longitudinal bars between stirrups and its diameter		4.375
A6 factor		620.0
C factor		0.5
RO factor		20.0
Ratio between the hardening stiffness in cyclic behavior and the elastic one		0.010
A1 factor		18.5
A2 factor		0.15
BETON_UNI (concrete) model	Unconfined concrete	Confined concrete
Young's modulus	0.30 10^5	0.2254 10^5
Poisson's coefficient	0.20	0.25
Maximum compression strength	30.0	30.0
Strain corresponding to 'STFC'	.002	.002
Maximum tension strength	3.0	3.0
Confinment parameter	.22687	.22687
Confinment parameter	.32912	.32912
Slope of the descending part of the compression curve	100.0	0.0
Stress defining the compression plateau	.0	6.0
Factor defining the softening behavior of the tension curve	3.0	3.0

Test plas14 **Results**

CASTEM FIGURES



Section:blue=steel,yellow=unconfined concrete,red=confined concre



curve: mocu (rouge) and takeda (vert)

```
*          Test Plas14.dgibi: Jeux de données
*          ---------------------------------
*
graph='N';
saut page;
opti echo 1;
*-------------------------------------------------------
*
*   EXEMPLE OF A CONCRETE SQUARE SECTION WITH 4 STEEL
*                      20mm BARS
*     CONCRETE DIMENSIONS: (0.25*0.25)m2 - ALL SECTION
*                          (0.20*0.20)m2 - CORE SECTION
*     TRANSVERSAL REINFORCEMENT: 10mm Bars / 7.0cm
*                      NOVEMBER 1993
*
*-------------------------------------------------------
*    Quadrangular Elements
*-------------------------------------------
   opti dime 2 elem qua4 echo 0 ;
*-------------------------------------------
*    Triangular Elements
*-------------------------------------------
*  opti dime 2 elem tri3;
*-------------------------------------------
*    DEFINITION OF THE STEEL GEOMETRY
*-------------------------------------------
ps1 = -9.386d-2  9.386d-2;
ps2 = -9.386d-2  7.614d-2;
ps3 =  1.772d-2   0.0    ;
cars1 = ps1 d 1 ps2 tran 1 ps3;
*
cars2 = cars1 plus ( 17.0e-2   0.0);
cars3 = cars2 plus ( 0.0  -17.0e-2);
cars4 = cars3 plus (-17.0e-2   0.0);
*
meshf = cars1 et cars2 et cars3 et cars4;
meshf = coul meshf bleu;
*-------------------------------------------
*    DEFINITION OF THE CONCRETE GEOMETRY
*-------------------------------------------
*        - UNCONFINED
*-------------------------------------------
*        verd - vertical number of divisions
*        hord - horizontal number of divisions
*-------------------------------------------
verd = 1;
hord = 8;
*
divu = 2;
*-------------------------------------------
pc1 = -12.5d-2  12.5d-2;
pc2 = -12.5d-2  10.0d-2;
pc3 =   2.5d-2   0.0d-2;
carc1 = pc1 d (divu) pc2 tran (divu) pc3;
*
carc2 = carc1 plus ( 22.5d-2   0.0);
carc3 = carc2 plus ( 0.0  -22.5d-2);
carc4 = carc3 plus (-22.5d-2   0.0);
*
pc4 = -10.0d-2  12.5d-2;
pc5 = -10.0d-2  10.0d-2;
pc6 =  20.0d-2   0.0d-2;
carc5 = pc4 d (divu) pc5 tran (verd) pc6;
*
carc6 = carc5 plus ( 0.0  -22.5d-2);
*
pc7 = -12.5d-2  10.0d-2;
pc8 = -10.0d-2  10.0d-2;
* pc9 =   0.0d-2 -20.0d-2;
* carc7 = pc7 d (divu) pc8 tran (hord) pc9;
* *
carc8 = carc7 plus ( 22.5d-2   0.0);
*
meshu = carc1 et carc2 et carc3 et carc4 et
        carc5 et carc6 et carc7 et carc8;
meshu = coul meshu jaune;
*-------------------------------------------
*        - CONFINED
*-------------------------------------------
pc10 = -10.0d-2  10.0d-2;
pc11 =  10.0d-2  10.0d-2;
pc12 =   0.0d-2 -20.0d-2;
meshc = pc10 d (verd) pc11 tran (hord) pc12;
meshc = coul meshc rouge;
*
elim (meshf et meshu et meshc) 0.001;
*
titre
'Section:blue=steel,yellow=unconfined concrete
,red=confined concrete';
*
si (ega graph 'Y');
trac (meshf et meshu et meshc);
finsi;
*
opti dime 3;
*
pp0 =   0.0  .0 .0;
pp1 =   1.0  .0 .0;
*
llb = pp0 d 1 pp1;
*
elim (llb et pp0 et pp1) 0.001;
*-------------------------------------------
*    CARACTERIZATION OF THE STEEL AND CONCRETE MODELS
*-------------------------------------------
*-------------------------------------------
*    Quadrangular Elements
*-------------------------------------------
   modf = modl meshf mecanique elastique PLASTIQUE
          ACIER_UNI quas;
   modu = modl meshu mecanique elastique PLASTIQUE
          BETON_UNI quas;
   modc = modl meshc mecanique elastique PLASTIQUE
          BETON_UNI quas;
*-------------------------------------------
*    Triangular Elements
*-------------------------------------------
*  modf = modl meshf mecanique elastique fibre_nl
*         ferraille tris;
*  modu = modl meshu mecanique elastique fibre_nl
*         beton tris;
*  modc = modl meshc mecanique elastique fibre_nl
*         beton tris;
*-------------------------------------------
*   Steel
*-------------------------------------------
matf = matr modf 'YOUN' 2.03e5 'NU'   0.30
'STSY' 440.0 'EPSU' .090   'STSU' 760.0  'EPSH' 0.030
'FALD' 4.375  'A6FA' 620.0 'CFAC'  0.5  'AFAC' 0.008
'ROFA' 20.0   'BFAC' 0.010 'A1FA'  18.5 'A2FA' 0.15;
carf = carb modf 'ALPY' 1.     'ALPZ' 1.;
*-------------------------------------------
*   Unconfined concrete
*-------------------------------------------
matu = matr modu 'YOUN' 0.30e5 'NU'   .20
```

```
'STFC' 30.0   'EZER' .002  'STFT' 3.0
'ALF1' .22687 'OME1' .32912  'ZETA' 100.0   'ST85' .0
'TRAF' 3.0 'FACL'  1. 'FAMX' 10. 'STPT' .0 'FAM1' 1.
'FAM2' 10.;
caru = carb modu 'ALPY' .66    'ALPZ' .66;
*-------------------------------------------
*   Confined concrete
*-------------------------------------------
*   Initial concrete Young modulus =
*     2 * STIFC / ( BETA * EZERO )
*-------------------------------------------
matc = matr modc 'YOUN' 0.2254e5 'NU' .25  'STFC' 30.0
'EZER' .002  'STFT' 3.0    'ALF1' .22687 'OME1' .32912
'ZETA' 0.0      'ST85' 6.0    'TRAF' 3.0
'FACL'  1. 'FAMX' 10. 'STPT' .0 'FAM1' 1. 'FAM2' 10.;
carc = carb modc 'ALPY' .66      'ALPZ' .66;
*
modq = modf et modu et modc;
macq = matf et matu et matc et carf et caru et carc;
*-------------------------------------------
*    USE OF "MOMCUR" PROCEDURE FOR THE ANALYSIS OF THE
*              PLASTIC BEHAVIOUR SECCION
*-------------------------------------------
*    CARACTERIZATION OF THE ACTION
*   (CURVATURES ALONG OY AXIS AND CONSTANT AXIAL FORCE)
*-------------------------------------------
eppl = 440.0/2.03e5;
*
cy = prog 0 pas .0005 .005 pas .005 .138;
*
ncur = dime cy;
cz = prog ncur *  .00;
fa = prog ncur * -.25;
*-------------------------------------------
*    RESOLUTION
*-------------------------------------------
my mz ea = mocu cy cz fa modq macq (1.d-6*eppl);
*-------------------------------------------
*    OUTPUT DIAGRAMS
*-------------------------------------------
c1= evol rouge manu 'Curvature' cy 'Moment'  (my*1.d3);
*-------------------------------------------
*    TRILINEAR CURVE FOR A TAKEDA MODEL
*          FOR THE SAME SECTION
*-------------------------------------------
abstak=prog 0. 2.03791E-03 1.85207E-02 1.38834E-01;
ordtak=prog 0. 2.05353E+01 7.10923E+01 7.06633E+01;
albnl=evol vert manu 'Curvature' abstak 'Moment' ordtak;
*-------------------------------------------
*    PLOT
*-------------------------------------------
si (ega graph 'Y');
  tt = table;
  tt.1 = 'MARQ CARR';
  tt.2 = '';
*
  titre 'courbe mocu (rouge) et takeda (vert)';
  dess (albnl et c1) tt;
*
finsi;
*-------------------------------------------
*    ERREUR
*-------------------------------------------
ordtak=ipol cy abstak ordtak;
errlis=ordtak - (my*1.d3);
errea=((ltl errlis errlis)**0.5) / (dime ordtak);
denom=((ltl ordtak ordtak)**0.5) / (dime ordtak);
errel=errea/denom;
mess 'erreur relative=' errel '(+-=3.5%)';
si (errel > 4.d-2); erre 5;
sinon;              erre 0; finsi;
fin;
```

Chapter 4

BEHAVIOUR LAWS

4.1 Test norton_cis1 Description sheet

Test name norton_cis1

Calculation type ISOTROPIC ELASTIC MECHANICS - 2D - PLANE STRAIN **Finite element type**

QUA4

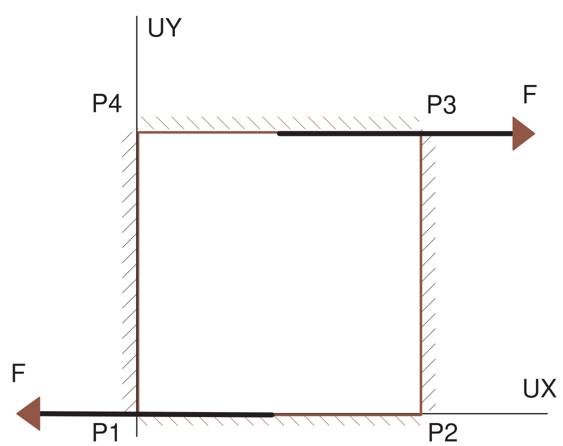
Topic Norton creep model for shear stresses.

The structure is a square plate subjected to shear stress (imposed load). It is embedded at each side. The plate follows Norton creep model. **Goal** Test the Norton model for solid plates.

Version 97' customer version

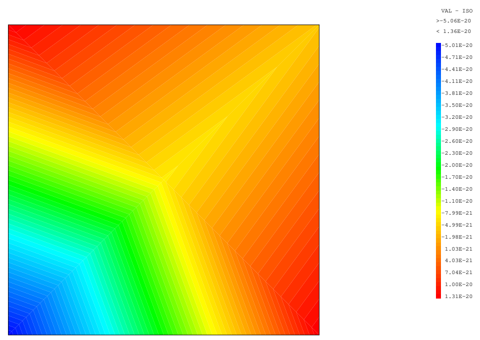
Model description

E	=	$2.1\ 10^{11}$
NU	=	0.3
AF1	=	$1.\ 10^{-45}$
AF2	=	5.
AF1	=	1.2
SMAX	=	$2.1\ 10^{8}$

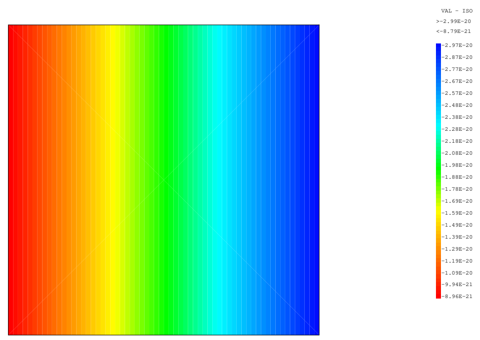


Test norton_cis1 Results

CASTEM FIGURES



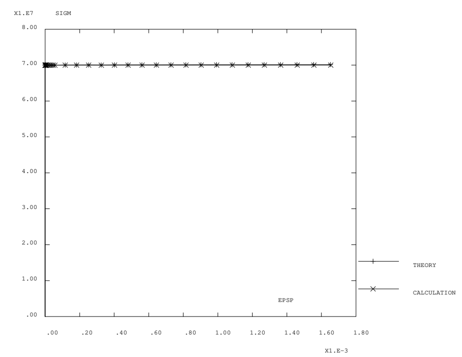
STRESSES IN THE X-DIRECTION



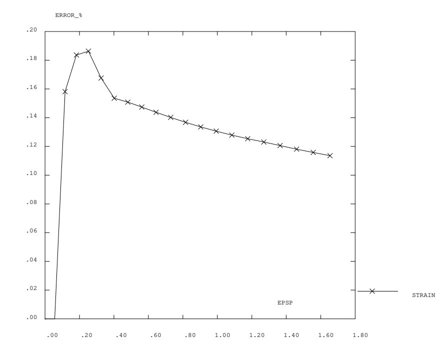
STRESSES IN THE Y-DIRECTION

SHEAR STRESSES

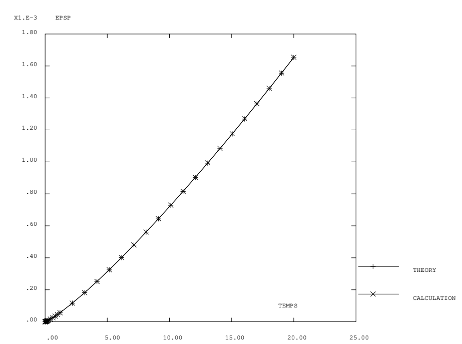
FINAL STRAIN



COMPARISON BETWEEN CALCULATION & THEORY



EVOLUTION OF ERRORS IN TERME OF %



COMPARISON BETWEEN CALCULATION THEORY, STRAIN-TIME

```
*       Test Norton_cis1.dgibi: Jeux de données
*       ---------------------------------------
*
*
**************************************************
*                                                *
*  TEST DE VALIDATION D'UNE LOI DE COMPORTEMENT  *
*  --------------------------------------------  *
*              DE MATERIAU                       *
*              -----------                       *
*                                                *
*  LOI DE COMPORTEMENT DE FLUAGE DE :            *
*              NORTON                            *
*                                                *
*  TEST POUR DES ELEMENTS MASSIFS                *
*  ------------------------------                *
*                                                *
*  MAILLAGE:                                     *
*   UNE PLAQUE CARREE DE COTE L=1 M              *
*                                                *
*  CHARGEMENT:                                   *
*   CISAILLEMENT PUR                             *
*   FORCES IMPOSEES                              *
*   DEFORMATIONS PLANES                          *
*                                                *
**************************************************
opti echo 0 dime 2 elem qua4 ;
*
*  Maillage
*
p1 = 0. 0. ;
p2 = 1. 0. ;
p3 = 1. 1. ;
p4 = 0. 1. ;
li1 = d p1 1 p2 ;
li2 = d p2 1 p3 ;
li3 = d p3 1 p4 ;
li4 = d p4 1 p1 ;
mail1 = 'SURF' ( li1 'ET' li2 'ET' li3 'ET' li4 ) 'PLAN' ;
*
*  Modele
*
AF1 = 1.E-45 ;
AF2 = 5. ;
AF3 = 1.2 ;
SMAX = 2.1E8 ;
youn0 = 2.1E11 ;
nu0 = .3 ;
mod0 = 'MODE' mail1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'FLUAGE' 'NORTON' ;
mat0 = 'MATE' mod0 'YOUN' youn0 'NU' nu0 'AF1' AF1
       'AF2' AF2 'AF3' AF3 'SMAX' SMAX ;
*
*  Conditions aux limites
*
cl1 = 'BLOQ' mail1 'UY' ;
cl2 = 'BLOQ' li1 'UX' ;
cl0 = cl1 'ET' cl2 ;
*
*  Chargement
*
depmax = 7.E7 ;
*chp1 = 'COOR' 2 mail1 ;
*chp1 = chp1 * depmax / 1. ;
*chp1 = 'NOMC' chp1 'UX' ;
*titr ' Champ de forces impose' ;
*trace chp1 mail1 ;
*chp2 = 'MANU' 'CHPO' mail1 1 'UY' 0. ;
* *chp0 = chp1 + chp2 ;
* dep1 = 'FORCE' li1 'FX' ( -1. * depmax ) 'FY' 0. ;
* dep2 = 'FORCE' li3 'FX' depmax 'FY' 0. ;
  dep0 = dep1 'ET' dep2 ;
  ev0 = 'EVOL' 'MANU' temps ( 'PROG' 0. 1.E-4 1000. )
  y ( 'PROG' 0. 1. 1. ) ;
  cha0 = 'CHAR' 'MECA' dep0 ev0 ;
  *
  dt0 = .020 ;
  dt1 = 2. * dt0 ;
  dt2 = 5. * dt1 ;
  dt3 = 5. * dt2 ;
  tfin0 = 20. ;
  *
  *  Appel à PASAPAS
  *
  ta1 = 'TABLE' ;
  ta1 .'MODELE' = mod0 ;
  ta1 .'CARACTERISTIQUES' = mat0 ;
  ta1 .'CHARGEMENT' = cha0 ;
  ta1 .'BLOCAGES_MECANIQUES' = cl0 ;
  ta1 .'TEMPS_CALCULES' = ( 'PROG' 0. 'PAS' 1.E-4 2.E-4 ) 'ET'
  ( 'PROG' 3.E-4 'PAS' dt0 dt0 ) 'ET'
  ( 'PROG' ( dt0 + dt0 ) 'PAS' dt0 dt1 ) 'ET'
  ( 'PROG' ( dt1 + dt0 ) 'PAS' dt1 dt2 ) 'ET'
  ( 'PROG' ( dt2 + dt1 ) 'PAS' dt2 dt3 ) 'ET'
  ( 'PROG' ( dt3 + dt2 ) 'PAS' dt3 tfin0 ) ;
  *
  PASAPAS ta1 ;
  *
  *  Post traitement
  *
  dim0 = 'DIME' ta1 .'TEMPS' ;
  eps10 = 'PROG' 0. ;
  sigm10 = 'PROG' 0. ;
  def10 = 'PROG' 0. ;
  def_th2 = 'PROG' 0. ;
  sig_th2 = 'PROG' 0. ;
  err_e2 = 'PROG' 0. ;
  err_s2 = 'PROG' 0. ;
  temp2 = 'PROG' 0. ;
  rac0 = ( 3. ) ** ( .5 ) ;
  H0 = youn0 / ( 2. * ( 1. + nu0 ) ) ;
  *
  prob0 = vrai ;
  'REPETER' bloc0 ( dim0 - 1 ) ;
    i1 = &bloc0 ;
  *
  *  Résultats calculés
  *
    def0 = ta1 .'DEFORMATIONS_INELASTIQUES'.i1 ;
    depl0 = ta1 .'DEPLACEMENTS'.i1 ;
    sigm0 = ta1 .'CONTRAINTES'.i1 ;
    temp0 = ta1 .'TEMPS'.i1 ;
    var0 = ta1 .'VARIABLES_INTERNES'.i1 ;
    eps0 = 'EPSI' mod0 depl0 ;
    eps1 = 'EXTR' ( 'EXCO' eps0 'GAXY' 'SCAL' ) 'SCAL' 1 1 1 ;
    sigm1 = 'EXTR' ( 'EXCO' sigm0 'SMXY' 'SCAL' ) 'SCAL' 1 1 1 ;
    def1 = 'EXTR' ( 'EXCO' def0 'GIXY' 'SCAL' ) 'SCAL' 1 1 1 ;
    eps10 = eps10 'ET' ( 'PROG' eps1 ) ;
    sigm10 = sigm10 'ET' ( 'PROG' sigm1 ) ;
    sigeq0 = 'VMIS' sigm0 mod0 ;
    sigeq1 = 'EXTR' sigeq0 'SCAL' 1 1 1 ;
    def10 = def10 'ET' ( 'PROG' def1 ) ;
    temp2 = temp2 'ET' ( 'PROG' temp0 ) ;
    var1 = 'EXTR' ( 'EXCO' var0 'EPSE' 'SCAL' ) 'SCAL' 1 1 1 ;
    sig0 = 'CHAN' 'CHPO' eps0 mod0 ;
    sigx = 'EXCO' sig0 'EPXX' ;
```

```
  sigy = 'EXCO' sig0 'EPYY' ;
  sigxy = 'EXCO' sig0 'GAXY' ;
*titr ' Contraintes suivant X' ;
*trace sigx mail1 ;
*titr ' Contraintes suivant Y' ;
*trace sigy mail1 ;
*titr ' Contraintes de cisaillement' ;
*trace sigxy mail1 ;
*
*  Résultats analytiques
*
y2 = ( ( rac0 * depmax ) ** ( AF2 ) ) ;
y2 = y2 * ( temp0 ** AF3 ) ;
x2 = y2 * AF1 * rac0 ;
*
def_th1 = x2 ;
sig_th1 = H0 * ( eps1 - x2 ) ;
*
def_th2 = def_th2 'ET' ( 'PROG' def_th1 ) ;
sig_th2 = sig_th2 'ET' ( 'PROG' sig_th1 ) ;
*mess ' sig calculée      =' sigm1 ;
*mess ' sig theorique     =' sig_th1 ;
*mess ' def calculée      =' def1 ;
*mess ' def théorique     =' def_th1 ;
*
*  Erreurs
*
'SI' ( def1 '>' ( 1.E-1 * eps1 ) ) ;
   err_e1 = ( 'ABS' ( def1 - def_th1 ) ) / def1 ;
   err_e1 = err_e1 * 100. ;
'SINON' ;
   err_e1 = 0. ;
'FINSI' ;
'SI' ( sigm1 '>' 0. ) ;
   err_s1 = ( 'ABS' ( sigm1 - sig_th1 ) ) / sigm1 ;
   err_s1 = err_s1 * 100. ;
'SINON' ;
   err_s1 = 0. ;
'FINSI' ;
*
'SI' ( ( err_e1 '>' .25 ) 'OU' ( err_s1 '>' .25 ) ) ;
   prob0 = faux ;
   mess 'err_e1 err_s1' err_e1 err_s1 ;
'FINSI' ;
err_e2 = err_e2 'ET' ( 'PROG' err_e1 ) ;
err_s2 = err_s2 'ET' ( 'PROG' err_s1 ) ;
*
'MESS' ' Fin du calcul de restitution au temps ' temp0 ;
*mess ' ' ;
*
'FIN' bloc0 ;
*
defo0 = 'DEFO' mail1 depl0 0. ;
defo1 = 'DEFO' mail1 depl0 1. 'ROUG' ;
*titr ' deformee finale' ;
*trace ( defo0 'ET' defo1 ) ;
*
'SI' ( prob0 'EGA' faux ) ;
   'ERRE' 5 ;
'SINON' ;
   'ERRE' 0 ;
'FINSI' ;
*
*  Tracés
*
evo1 = 'EVOL' 'MANU' 'epsp' def10 'sigm' sigm10 ;
evo2 = 'EVOL' 'MANU' 'epsp' def_th2 'sigm' sig_th2 ;
evo3 = 'EVOL' 'MANU' 'epsp' def10 'erreur_%' err_e2 ;
evo4 = 'EVOL' 'MANU' 'epsp' def10 'erreur_%' err_s2 ;
evo5 = 'EVOL' 'MANU' 'temps' temp2 'epsp' def10 ;
evo6 = 'EVOL' 'MANU' 'temps' temp2 'epsp' def_th2 ;
TAB = 'TABLE' ;
TAB. 1 = 'MARQ CROI ' ;
TAB. 2 = 'MARQ PLUS ' ;
TAB.'TITRE' = 'TABLE' ;
TAB.'TITRE'. 1 = MOT 'CALCUL' ;
TAB.'TITRE'. 2 = MOT 'THEORIE' ;
*'DESSIN' ( evo1 'ET' evo2 )
*'TITR' ' Comparaison calcul theorie'
*'LEGE' TAB ;
TAB = 'TABLE' ;
TAB. 1 = 'MARQ CROI ' ;
*TAB. 2 = 'MARQ PLUS ' ;
TAB.'TITRE' = 'TABLE' ;
TAB.'TITRE'. 1 = MOT 'DEFORMATION' ;
*TAB.'TITRE'. 2 = MOT 'CONTRAINTE' ;
*'DESSIN' ( evo3 )
*'TITR' ' Evolution des erreurs (en pourcentage) '
*'LEGE' TAB ;
TAB = 'TABLE' ;
TAB. 1 = 'MARQ CROI ' ;
TAB. 2 = 'MARQ PLUS ' ;
TAB.'TITRE' = 'TABLE' ;
TAB.'TITRE'. 1 = MOT 'CALCUL' ;
TAB.'TITRE'. 2 = MOT 'THEORIE' ;
*'DESSIN' ( evo5 'ET' evo6 )
*'TITR' ' Comparaison calcul theorie, deformation-temps'
*'LEGE' TAB ;
*
'FIN' ;
```

Test norton_cis1 **Comments**

- The Norton's creep model

```
MODO = MODE MAIL1 MECANIQUE ELASTIQUE ISOTROPE
       FLUAGE NORTON ;
MATO = MATE MODO YOUN YOUNO NU NUO 'AF1' AF1
       'AF2' AF2 'AF3' AF3 'SMAX' SMAX ;
```

Several creep models are available in CASTEM2000. In each case we have to specify the elastic behavio,r the equivalent creep deformation (ef), the equivalent stress (s) and the time (t).
The parameters to be input in Norton's creep model are AF1, AF2, AF3, as well as a SMAX stress reference (equal to Young's modulus times 1.E-3 by default).
The constitutive equation Norton creep model is:

$$\boxed{\text{ef} = \text{AF1 * (s**AF2) * (t**AF3)}}$$

4.2 Test chaboche2 Description sheet

Test name chaboche2

Calculation type ISOTROPIC VISCOPLASTIC MECHANICS - 2D AXIS **Finite element type** COQ8

Topic Unified viscoplastic model of Chaboche

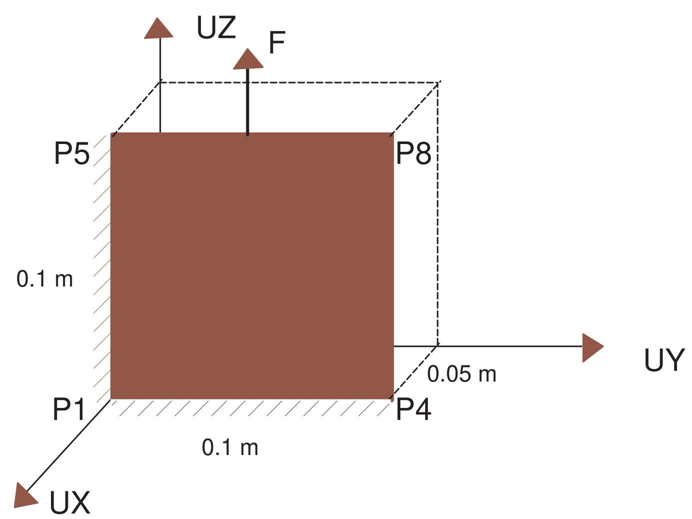
The structure is an embedded square plate. It is subjected to imposed displacements at the upper surface. The plate follows the Chaboche's viscoplastic law model.

Goal Test the Chaboche model for solid structure.

Version 97' customer version

Model description

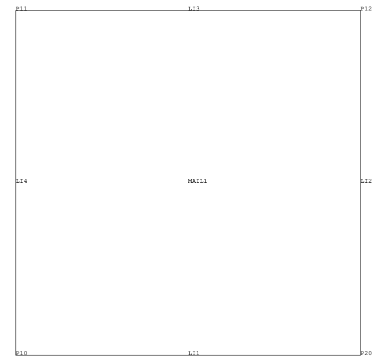
E	=	$7.34\ 10^{10}$
NU	=	0.33
RHO	=	$7.8\ 10^{3}$
ALPHA	=	$1.\ 10^{-5}$



Test chaboche2

Results

CASTEM FIGURES



MAILLAGE

SHEAR STRESS

REACTIONS

STRESS IN THE Y DIRECTION

DEFORMATION

STRESS IN THE Z DIRECTION

```
*       Test Chaboche2.dgibi: Jeux de données
*       ------------------------------------
*
complete = faux;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
***************************************************
**************************************************
*                                                *
*  TEST DE VALIDATION D'UNE LOI DE COMPORTEMENT  *
*  --------------------------------------------  *
*            DE MATERIAU                         *
*            -----------                         *
*                                                *
*  LOI DE COMPORTEMENT VISCOPLASTIQUE DE:        *
*         CHABOCHE                               *
*                                                *
*  COMPARAISON DE CALCULS SUR DES ELEMENTS:      *
*     - COQUE EPAISSE ( MFR=5 )                  *
*     - COQUE MINCE   ( MFR=3 )                  *
*     - MASSIF 3D     ( MFR=1 )                  *
*                                                *
*  STRUCTURE : UNE PLAQUE CARREE                 *
*     - COTE : .1 M                              *
*     - EPAISSEUR : 5.E-4 M                      *
*                                                *
*  CHARGEMENT : DEPLACEMENTS IMPOSES             *
*             ESSAI DE TRACTION                  *
*                                                *
**************************************************
opti echo 0 dime 3 elem qua8 ;
*
*
*  Maillage
*
epais0 = .0005 ;
rayon0 = .05 ;
l10 = .1 ;
n10 = 1 ;
p10 = rayon0 0. 0. ;
p20 = rayon0 l10 0. ;
p11 = rayon0 0. l10 ;
p12 = rayon0 l10 l10 ;
li1 = d p10 n10 p20 ;
li2 = d p20 n10 p12 ;
li3 = d p12 n10 p11 ;
li4 = d p11 n10 p10 ;
mail1='SURF' ( li1 'ET' li2 'ET' li3 'ET' li4 ) 'PLAN';
*titr 'maillage' ;
*trace mail1 ;
*
*    Modele de calcul
*
mod00 = 'MODE' mail1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'VISCOPLASTIQUE' 'CHABOCHE' 'COQ8' ;
mat00 = 'MATE' mod00 'YOUN' 7.34E10 'NU' 0.33
       'RHO' 7.8E3 'N' 24
       'KK' 10.E6 'K0' 116.E6 'ALFK' 1.5 'ALFR' 0.35
       'ALF' 2.E6 'A1' 67.5E6 'C1' 1300 'BET1' 4807.E6
       'R1' 4  'A2' 80.E6 'C2' 45 'BET2' 58480.E6
       'R2' 4 'PHI' 1.  'B' 12 'GAMA' 2.E-7 'M' 2
       'QMAX' 455.E6 'QSTA' 200.E6 'MU' 19 'ETA' 0.50 ;
car00 = 'CARA' mod00 'EPAI' epais0 ;
*
*    Conditions aux limites
*
cl10 = 'BLOQ' mail1 'UX' ;
* cl20 = 'BLOQ' li1 'UZ' ;
* cl30 = 'BLOQ' li3 'UZ' ;
* cl40 = 'BLOQ' li4 'UY' ;
  cl00 = cl10 'ET' cl20 'ET' cl30 'ET' cl40 ;
  *
  *    Chargement
  *
  depmax = .0050 ;
  dep10 = 'DEPI' cl30 depmax ;
  ev00 = 'EVOL' 'MANU' temps ( 'PROG' 0. 1000. ) y
         ( 'PROG' 0. 1. ) ;
  cha00 = 'CHAR' 'DIMP' dep10 ev00 ;
  *vec1 = 'VECT' dep10 1.E-5 'FX' 'FY' 'FZ' 'ROUG' ;
  *titr ' chargement ' ;
  *trace vec1 mail1 ;
  *
  *  Temps du calcul
  *
  dt00 = 5. ;
  si complet;
  tfin00 = 200. ;
  sinon;
  tfin00 = 25.;
  finsi;
  *
  *
  *  Resolution par PASAPAS
  *--------------------------------------
  *
  ta10 = 'TABLE' ;
  ta10 .'MODELE' = mod00 ;
  ta10 .'CARACTERISTIQUES' = mat00 'ET' car00 ;
  ta10 .'BLOCAGES_MECANIQUES' = cl00 ;
  ta10 .'CHARGEMENT' = cha00 ;
  ta10 .'TEMPS_CALCULES' =  'PROG' 0. 'PAS' dt00 tfin00 ;
  *( 'PROG' 22. 'PAS' 2. 40. ) 'ET'
  *( 'PROG' 41. 'PAS' dt00 tfin00 ) ) ;
  *
  PASAPAS ta10 ;
  *
  *
  *  Post-traitement
  *----------------------------------------
  *
  dim10 = 'DIME' ta10 .'TEMPS' ;
  depl120 = 'PROG' 0. ;
  *
  reac0 = ta10 .'REACTIONS'.( dim10 - 1 ) ;
  depl0 = ta10 .'DEPLACEMENTS'.( dim10 - 1 ) ;
  vec2 = 'VECT' reac0 1.E-2 'FX' 'FY' 'FZ' 'ROUG' ;
  *titr ' reactions ' ;
  *trace vec2 mail1 ;
  defo0 = 'DEFO' mail1 depl0 0. ;
  defo1 = 'DEFO' mail1 depl0 1. 'ROUG' ;
  titr ' deformees ' ;
  *trace ( defo0 'ET' defo1 ) ;
  *
  'REPETER' bloc0 ( dim10 - 1 ) ;
    i10 = &bloc0 ;
    reac0 = ta10 .'REACTIONS'.i10 ;
    depl10 = ta10 .'DEPLACEMENTS'.i10 ;
    temp10 = ta10 .'TEMPS'.i10 ;
    depl11 = 'EXTR' depl10 'UY' p11 ;
    sigm0 = ta10 .'CONTRAINTES'.i10 ;
    sigm0 = 'RTENS' sigm0 mod00 ( 0 1 0 ) car00 ;
    sigm1 = 'CHAN' 'CHPO' sigm0 mod00 ;
   'SI' ( i10 'EGA' ( dim10 - 1 ) ) ;
    titr ' Contrainte de cisaillement ' ;
```

```
*  trace ( 'EXCO' sigm1 'SMST' ) mail1 ;
  titr ' Contrainte suivant Y' ;
*  trace ( 'EXCO' sigm1 'SMSS' ) mail1 ;
  titr ' Contrainte suivant Z ' ;
*  trace ( 'EXCO' sigm1 'SMTT' ) mail1 ;
 'FINSI' ;
  reac0 = 'REDU' reac0 li3 ;
  reac0 = 'EXCO' reac0 'FZ' 'SCAL' ;
  chp1 = 'MANU' 'CHPO' li3 1 'SCAL' 1. ;
  depl11 = 'XTY' reac0 chp1 ( 'MOTS' 'SCAL' )
          ( 'MOTS' 'SCAL' ) ;
  depl120 = depl120 'ET' ( 'PROG' depl11 ) ;
'FIN' bloc0 ;
*
*
opti echo 0 dime 3 elem tri3 ;
*
*
*  Maillage
*
epais0 = .0005 ;
rayon0 = .05 ;
l10 = .1 ;
n10 = 1 ;
p10 = rayon0 0. 0. ;
p20 = rayon0 l10 0. ;
p11 = rayon0 0. l10 ;
p12 = rayon0 l10 l10 ;
li1 = d p10 n10 p20 ;
li2 = d p20 n10 p12 ;
li3 = d p12 n10 p11 ;
li4 = d p11 n10 p10 ;
mail1='SURF' ( li1 'ET' li2 'ET' li3 'ET' li4 ) 'PLAN';
*titr 'maillage' ;
*trace mail1 ;
*
*    Modele de calcul
*
mod00 = 'MODE' mail1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'VISCOPLASTIQUE' 'CHABOCHE' 'DKT' ;
mat00 = 'MATE' mod00 'YOUN' 7.34E10 'NU' 0.33
       'RHO' 7.8E3 'N' 24
       'KK' 10.E6 'K0' 116.E6 'ALFK' 1.5 'ALFR' 0.35
       'ALF' 2.E6 'A1' 67.5E6 'C1' 1300 'BET1' 4807.E6
       'R1' 4 'A2' 80.E6 'C2' 45 'BET2' 58480.E6
       'R2' 4 'PHI' 1. 'B' 12 'GAMA' 2.E-7 'M' 2
       'QMAX' 455.E6 'QSTA' 200.E6 'MU' 19 'ETA' 0.50 ;
car00 = 'CARA' mod00 'EPAI' epais0 'ALFA' ( 2. / 3. ) ;
*
*    Conditions aux limites
*
cl10 = 'BLOQ' mail1 'UX' ;
cl20 = 'BLOQ' li1 'UZ' ;
cl30 = 'BLOQ' li3 'UZ' ;
cl40 = 'BLOQ' li4 'UY' ;
cl00 = cl10 'ET' cl20 'ET' cl30 'ET' cl40 ;
*
*    Chargement
*
depmax = .0050 ;
dep10 = 'DEPI' cl30 depmax ;
ev00 = 'EVOL' 'MANU' temps ( 'PROG' 0. 1000. ) y
       ( 'PROG' 0. 1. ) ;
cha00 = 'CHAR' 'DIMP' dep10 ev00 ;
*vec1 = 'VECT' dep10 1.E-5 'FX' 'FY' 'FZ' 'ROUG' ;
*titr ' chargement ' ;
*trace vec1 mail1 ;
*
*  Temps du calcul
*
*
*
*  Resolution par PASAPAS
*--------------------------------------
*
ta10 = 'TABLE' ;
ta10 .'MODELE' = mod00 ;
ta10 .'CARACTERISTIQUES' = mat00 'ET' car00 ;
ta10 .'BLOCAGES_MECANIQUES' = cl00 ;
ta10 .'CHARGEMENT' = cha00 ;
ta10 .'TEMPS_CALCULES' =  'PROG' 0. 'PAS' dt00 tfin00 ;
*( 'PROG' 22. 'PAS' 2. 40. ) 'ET'
*( 'PROG' 41. 'PAS' dt00 tfin00 ) ) ;
*
PASAPAS ta10 ;
*
*
*  Post-traitement
*----------------------------------------
*
dim10 = 'DIME' ta10 .'TEMPS' ;
depl12 = 'PROG' 0. ;
var2 = 'PROG' 0. ;
*
reac0 = ta10 .'REACTIONS'.( dim10 - 1 ) ;
depl0 = ta10 .'DEPLACEMENTS'.( dim10 - 1 ) ;
vec2 = 'VECT' reac0 1.E-2 'FX' 'FY' 'FZ' 'ROUG' ;
*titr ' reactions ' ;
*trace vec2 mail1 ;
defo0 = 'DEFO' mail1 depl0 0. ;
defo1 = 'DEFO' mail1 depl0 1. 'ROUG' ;
titr ' deformees ' ;
*trace ( defo0 'ET' defo1 ) ;
*
'REPETER' bloc0 ( dim10 - 1 ) ;
  i10 = &bloc0 ;
  reac0 = ta10 .'REACTIONS'.i10 ;
  depl10 = ta10 .'DEPLACEMENTS'.i10 ;
  temp10 = ta10 .'TEMPS'.i10 ;
  sigm0 = ta10 .'CONTRAINTES'.i10 ;
  def0 = ta10 .'DEFORMATIONS_INELASTIQUES'.i10 ;
  var0 = ta10. 'VARIABLES_INTERNES'.i10 ;
*  var1 = 'EXTR' ( 'EXCO' var0 'GPN3' ) p11 ;
  eps0 = 'EPSI' depl10 mod00 car00 ;
  sigm0 = 'RTENS' sigm0 mod00 ( 0 1 0 ) ;
  eps0 = 'RTENS' eps0 mod00 ( 0 1 0 ) ;
  sigm1 = 'CHAN' 'CHPO' sigm0 mod00 ;
  eps1 = 'CHAN' 'CHPO' eps0 mod00 ;
  def0 = 'RTENS' def0 mod00 ( 0 1 0 ) ;
  def1 = 'CHAN' 'CHPO' def0 mod00 ;
 'SI' ( i10 'EGA' ( dim10 - 1 ) ) ;
  titr ' Contrainte de cisaillement ' ;
*  trace ( 'EXCO' sigm1 'N12' ) mail1 ;
  titr ' Contrainte suivant Y' ;
*  trace ( 'EXCO' sigm1 'N11' ) mail1 ;
  titr ' Contrainte suivant Z ' ;
*  trace ( 'EXCO' sigm1 'N22' ) mail1 ;
  titr ' déformation plastique de cisaillement ' ;
*  trace ( 'EXCO' def1 'GAST' ) mail1 ;
  titr ' déformation plastique suivant Y' ;
*  trace ( 'EXCO' def1 'EPSS' ) mail1 ;
  titr ' déformation plastique Z ' ;
*  trace ( 'EXCO' def1 'EPTT' ) mail1 ;
  titr ' déformation de cisaillement ' ;
*  trace ( 'EXCO' eps1 'GAST' ) mail1 ;
  titr ' déformation suivant Y' ;
```

```
*  trace ( 'EXCO' eps1 'EPSS' ) mail1 ;
  titr ' déformation suivant Z ' ;
*  trace ( 'EXCO' eps1 'EPTT' ) mail1 ;
 'FINSI' ;
  depl11 = 'EXTR' depl10 'UZ' p11 ;
  reac0 = 'REDU' reac0 li3 ;
  reac0 = 'EXCO' reac0 'FZ' 'SCAL' ;
  chp1 = 'MANU' 'CHPO' li3 1 'SCAL' 1. ;
  depl11 = 'XTY' reac0 chp1 ( 'MOTS' 'SCAL' )
           ( 'MOTS' 'SCAL' ) ;
  depl12 = depl12 'ET' ( 'PROG' depl11 ) ;
*  var2 = var2 'ET' ( 'PROG' var1 ) ;
'FIN' bloc0 ;
*
*
opti echo 0 dime 3 elem cub8 ;
*
*
*  Maillage
*
rayon0 = .05 ;
epais0 = .0005 ;
l1 = epais0 + rayon0 ;
l2 = .1 ;
n1 = 1 ;
n2 = 1 ;
p1 = rayon0 0. 0. ;
p2 = l1 0. 0. ;
p3 = l1 l2 0. ;
p4 = rayon0 l2 0. ;
p11 = rayon0 0. l2 ;
p21 = l1 0. l2 ;
p31 = l1 l2 l2 ;
p41 = rayon0 l2 l2 ;
d1 = d p1 n1 p2 ;
d2 = d p2 n2 p3 ;
d3 = d p3 n1 p4 ;
d4 = d p4 n2 p1 ;
d11 = d p11 n1 p21 ;
d21 = d p21 n2 p31 ;
d31 = d p31 n1 p41 ;
d41 = d p41 n2 p11 ;
p01 = ( rayon0 + ( epais0 / 2. ) ) 0. l2 ;
p02 = ( rayon0 + ( epais0 / 2. ) ) l2 l2 ;
sur1 = 'DALL' d1 d2 d3 d4 ;
sur2 = 'DALL' d11 d21 d31 d41 ;
vol1 = sur1 'VOLU' n2 sur2 ;
*titr ' maillage ' ;
*trace vol1 ;
*
*    Modele de calcul
*
mod0 = 'MODE' vol1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'VISCOPLASTIQUE' 'CHABOCHE' ;
mat0 = 'MATE' mod0 'YOUN' 7.34E10 'NU' 0.33
       'RHO' 7.8E3 'ALPHA' 1.E-5 'N' 24
       'KK' 10.E6 'K0' 116.E6 'ALFK' 1.5 'ALFR' 0.35
       'ALF' 2.E6  'A1' 67.5E6 'C1' 1300 'BET1' 4807.E6
       'A2' 80.E6 'C2' 45 'BET2' 58480.E6 'R2' 4
       'PHI' 1. 'R1' 4  'B' 12 'GAMA' 2.E-7 'M' 2
       'QMAX' 455.E6 'QSTA' 200.E6 'MU' 19 'ETA' 0.50 ;
*
*    Conditions aux limites
*
cl1 = 'BLOQ' ( d2 'ET' d21 ) 'UX' ;
cl2 = 'BLOQ' sur1 'UZ' ;
cl3 = 'BLOQ' sur2 'UZ' ;
cl4 = 'BLOQ' ( d1 'ET' d11 ) 'UY' ;
```

```
cl0 = cl1 'ET' cl2 'ET' cl3 'ET' cl4 ;
*
*    Chargement
*
depmax = .0050 ;
dep1 = 'DEPI' cl3 depmax ;
ev0 = 'EVOL' 'MANU' temps ( 'PROG' 0. 1000. ) y
      ( 'PROG' 0. 1. ) ;
cha0 = 'CHAR' 'DIMP' dep1 ev0 ;
*
*  Temps du calcul
*
dt0 = 5. ;
si complet;
tfin0 = 200. ;
sinon;
tfin0 = 25. ;
finsi;
*
*
*  Resolution par PASAPAS
*----------------------------------------
*
ta1 = 'TABLE' ;
ta1 .'MODELE' = mod0 ;
ta1 .'CARACTERISTIQUES' = mat0 ;
ta1 .'BLOCAGES_MECANIQUES' = cl0 ;
ta1 .'CHARGEMENT' = cha0 ;
ta1 .'TEMPS_CALCULES' =  'PROG' 0. 'PAS' dt0 tfin0 ;
*( 'PROG' 22. 'PAS' 2. 40. ) 'ET'
*( 'PROG' 41. 'PAS' dt0 tfin0 ) ) ;
*
PASAPAS ta1 ;
*
*
*  Post-traitement
*------------------------------------------
*
dim0 = 'DIME' ( ta1 .'TEMPS' ) ;
char2 = 'PROG' 0. ;
depl2 = 'PROG' 0. ;
defi2 = 'PROG' 0. ;
var10 = 'PROG' 0. ;
*
reac0 = ta1 .'REACTIONS'.( dim0 - 1 ) ;
depl0 = ta1 .'DEPLACEMENTS'.( dim0 - 1 ) ;
vec2 = 'VECT' reac0 1.E-2 'FX' 'FY' 'FZ' 'ROUG' ;
*titr ' reactions ' ;
*trace vec2 vol1 ;
defo0 = 'DEFO' vol1 depl0 0. 'ROUG' ;
defo1 = 'DEFO' vol1 depl0 1. ;
titr ' deformees ' ;
*trace ( defo0 'ET' defo1 ) ;
reac1 = 'REDU' reac0 p1 ;
reac2 = 'REDU' reac0 p2 ;
vec01 = 'VECT' reac1 1.E-2 'FX' 'FY' 'FZ' 'ROUG' ;
vec02 = 'VECT' reac2 1.E-2 'FX' 'FY' 'FZ' 'VERT' ;
*titr ' reactions 2 ' ;
*trace ( vec01 'ET' vec02 ) vol1 ;
*
'REPETER' bloc0 ( dim0 - 1 ) ;
  i0 = &bloc0 ;
  reac0 = ta1 .'REACTIONS'.i0 ;
  depl0 = ta1 .'DEPLACEMENTS'.i0 ;
  sigm0 = ta1 .'CONTRAINTES'.i0 ;
  temp0 = ta1 .'TEMPS'.i0 ;
  def0 = ta1 .'DEFORMATIONS_INELASTIQUES'.i0 ;
  char1 = 'EXTR' depl0 'UZ' p21 ;
```

```
  reac0 = 'REDU' reac0 sur2 ;
  reac0 = 'EXCO' reac0 'FZ' 'SCAL' ;
  chp1 = 'MANU' 'CHPO' sur2 1 'SCAL' 1. ;
  depl11 = 'XTY' reac0 chp1 ( 'MOTS' 'SCAL' )
           ( 'MOTS' 'SCAL' ) ;
  depl2 = depl2 'ET' ( 'PROG' depl11 ) ;
  sigm0 = 'CHAN' 'CHPO' sigm0 mod0 ;
  def1 = 'CHAN' 'CHPO' def0 mod0 ;
  var0 = ta1 .'VARIABLES_INTERNES'.i0 ;
*  var1 = 'EXTR' ( 'EXCO' var0 'GPYZ' ) p21 ;
 'SI' ( i0 'EGA' ( dim0 - 1 ) ) ;
  titr ' déformation plastique de cisaillement ' ;
*  trace ( 'EXCO' def1 'GAYZ' ) vol1 ;
  titr ' déformation plastique suivant Y' ;
*  trace ( 'EXCO' def1 'EPYY' ) vol1 ;
  titr ' déformation plastique Z ' ;
*  trace ( 'EXCO' def1 'EPZZ' ) vol1 ;
  titr ' déformation plastique suivant X' ;
*  trace ( 'EXCO' def1 'EPXX' ) vol1 ;
  titr ' Contrainte de cisaillement' ;
*  trace ( 'EXCO' sigm0 'SMYZ' ) vol1 ;
  titr ' Contrainte suivant Y' ;
*  trace ( 'EXCO' sigm0 'SMYY' ) vol1 ;
  titr ' Contrainte suivant Z' ;
*  trace ( 'EXCO' sigm0 'SMZZ' ) vol1 ;
  titr ' Contrainte suivant X' ;
*  trace ( 'EXCO' sigm0 'SMXX' ) vol1 ;
 'FINSI' ;
*  cont0 =  'MAXI' ( 'EXCO' 'SMXX' sigm0 ) ;
*  cont1 = cont1 'ET' ( 'PROG' cont0 ) ;
  char2 = char2 'ET' ( 'PROG' char1 ) ;
*  var10 = var10 'ET' ( 'PROG' var1 ) ;
'FIN' bloc0 ;
*
*
opti echo 0 dime 3 ;
*
*
*  Données du calcul coque
*
*  Erreur
*
*
err_e1 = 'PROG' 0. ;
err_e2 = 'PROG' 0. ;
dim10 = 'DIME' char2 ;
*
bool0 = faux ;
'REPETER' bloc0 ( dim10 - 1 ) ;
  i10 = &bloc0 ;
  depl1 = 'EXTR' depl2 i10 ;
  depl11 = 'EXTR' depl12 i10 ;
  depl111 = 'EXTR' depl120 i10 ;
*
'SI' (depl1 'NEG' 0. ) ;
  err_e0 = 'ABS' ( ( depl1 - depl11 ) / depl1 ) ;
  err_e0 = err_e0 * 100. ;
  err_e00 = 'ABS' ( ( depl1 - depl111 ) / depl1 ) ;
  err_e00 = err_e00 * 100. ;
'SINON' ;
  err_e0 = 0. ;
  err_e00 = 0. ;
'FINSI' ;
*
'SI' ( ( err_e0 '>' 5.E-2 ) 'OU'
     ( err_e00 '>' 5.E-2 ) ) ;
   bool0 = vrai ;
'FINSI' ;
err_e1 = err_e1 'ET' ( 'PROG' err_e0 ) ;
err_e2 = err_e2 'ET' ( 'PROG' err_e00 ) ;
*
'FIN' bloc0 ;
*
'SI' ( bool0 'EGA' vrai ) ;
     'ERRE' 5 ;
'SINON' ;
     'ERRE' 0 ;
'FINSI' ;
*
*  Traces
*
evo1 = 'EVOL' 'MANU' 'depl' char2 'force' depl2 ;
evo2 = 'EVOL' 'MANU' 'depl' char2 'force' depl12 ;
evo3 = 'EVOL' 'MANU' 'chargement' char2 'erreur_%'
       err_e1 ;
evo4 = 'EVOL' 'MANU' 'depl' char2 'chargement' depl120;
evo5 = 'EVOL' 'MANU' 'chargement' char2 'erreur_%'
       err_e2 ;
TAB = 'TABLE' ;
TAB. 1 = 'MARQ PLUS ' ;
TAB. 2 = 'MARQ CROI ' ;
TAB. 3 = 'MARQ CARR ' ;
TAB.'TITRE'= 'TABLE' ;
TAB.'TITRE'. 1 = MOT 'MASSIF' ;
TAB.'TITRE'. 2 = MOT 'COQUE' ;
TAB.'TITRE'. 3 = MOT 'EPAISSE' ;
*'DESS' ( evo1 'ET' evo2 'ET' evo4 )
*'TITR' ' Deplacement de l extremite du cylindre'
*'LEGE' TAB ;
*
TAB = 'TABLE' ;
TAB. 1 = 'MARQ PLUS ' ;
TAB. 2 = 'MARQ CROI ' ;
TAB.'TITRE' = 'TABLE' ;
TAB.'TITRE'. 1 = MOT 'MINCE' ;
TAB.'TITRE'. 2 = MOT 'EPAISSE' ;
*'DESS' ( evo3 'ET' evo5 )
*'TITR' 'Erreur sur les deplacements '
*'LEGE' TAB ;
*
*
'FIN' ;
```

Test chaboche2 **Comments**

- The Chaboche's viscoplastic model

```
MODOO = MODE MAIL1 MECANIQUE ELASTIQUE ISOTROPE
        VISCOPLASTIQUE CHABOCHE COQ8 ;
MATOO = MATE MODOO YOUN 7.34E10 NU 0.33
        RHO 7.8E3 N 24
        KK 10.E6  K0 116.E6
        ALFK 1.5  ALFR 0.35 ALF 2.E6
        A1 67.5E6 C1 1300 BET1 4807.E6  R1 4
        A2 80.E6  C2 45  BET2 58480.E6 R2 4 PHI 1.
        B 12      GAMA 2.E-7 M 2
        QMAX 455.E6 QSTA 200.E6 MU 19 ETA 0.50 ;
```

The equations for the Chaboche's model are as follows:

-> *Criterion:* F = J2 (S-X) - RR - KK:

-> *Strain hardening:* dXi =2/3 * Ai*Ci*dEP - Ci*Xi*p'*dp
- (J2(Xi)/BETi)**(Ri-1) * Xi * dt (i=1 or 2)
if BETi is non zero, else:
dXi =2/3 * Ai*Ci*dEP - Ci*Xi*p'*dp' (i=1 or 2)
with : p' = PHI+(1-PHI)* e** (-B*p)
dp = < F/K >**N * e**(ALF* < F/K >**(N+1))
dR = B*(Q - R)*dp + GAMA*Sign(QR-R)*|QR-R|**M *dt

-> *Flow rule:* dEP = 3/2 * dp * < S-X > / J2(S-X)

-> *Memory of inelastic strain* G = I2 (EP-Y) - q < 0
dQ = 2*MU*(QMAX-Q)* dq
dq = ETA * < n.nn > * dp
dY = (3/2 ** 0.5) * (1-ETA) * < n.nn > * nn*dp

QR = Q - QSTA * (1 - ((QMAX-Q)/QMAX)**2)

The initial value of Q is Q0 and must be initialized: Q = Q0 = 30 MPa (steel 316). For this purpose, a field of internal variables must be created, with a component named 'QQ' , its value being Q0. This field will be passed into the table of PASAPAS.

The parameter values for steel 316L at 600C are given for information only. The data to be input are the following:

			Values for Steel 316
Threshold evolution law:			− − − − − − − − − −
'N '	:	viscoplasticity threshold exponent	24
'KK '	:	initial value for elastic limit	10 MPa
'K0 '	:	initial factor for the threshold normalization	116 MPa
'ALFK'	:	K isotropic evolution coefficient	1.5
'ALFR'	:	threshold isotropic evolution coefficient	0.35
'ALF '	:	viscosity coefficient	2.E6

		Values for Steel 316
Evolution law for the centres X1 and X2:		−−−−−−−−−−
'A1 ' :	plastic strain coefficient	67.5 MPa
'C1 ' :	reference term coefficient	1300
'BET1' :	normalization factor for restoration	4807 MPa
'R1 ' :	restoration term exponent	4
'A2 ' :	plastic strain coefficient	80 MPa
'C2 ' :	reference term coefficient	45
'BET2' :	normalization factor for restoration	5840 MPa
'R2 ' :	restoration term exponent	4
'PHI ' :	reference term multiplying coefficient	1.
Evolution law for isotropic stress hardening:		
'B ' :	isotropic stress hardening coefficient	12
'GAMA' :	restoration effect coefficient	2.E-7
'M ' :	restoration term exponent	2
Evolution law for the plastic deformation memory:		
'QMAX' :	Q maximal value	455 MPa
'QSTA' :	Q stabilized value	200 MPa
'MU ' :	Q evolution law coefficient	19
'ETA ' :	factor linking q with the plastic strain	0.06

The following parameters will have to be specified with the elastic behavior.

−> *Notation:* S stress tensor
Xi kinematic stress hardening variables (i=1 ou 2)
EP inelastic strain tensor
p accumulated equivalent inelastic strain
q isotropic variable for the memory surface in deformation
Y kinematic variable for the memory surface in deformation
I2 tensor second invariant
J2 deviatoric tensor second invariant
n normal from F criterion
nn normal from G threshold
t time
$< a >$ positive part of a
x.y scalar product of x by y
X = X1 for a single centre
X1+X2 for two centres
RR = ALFR * R
K = K0 + ALFK * R
Q(0)=Q0

For further details, see D. NOUAILHAS's report:
"A viscoplastic modelling applied to stainless steel behavior", Second Inter. Conf. on Constitutive Laws for Engineering Materials,
University of Arizona, Tucson,1987

4.3 Test ohno2 Description sheet

Test name ohno2

Calculation type ISOTROPIC VISCOPLASTIC MECHANICS - 2D AXIS **Finite element type** QUA8

Topic Ohno -modified Chaboche-model for viscoplastic behaviour.

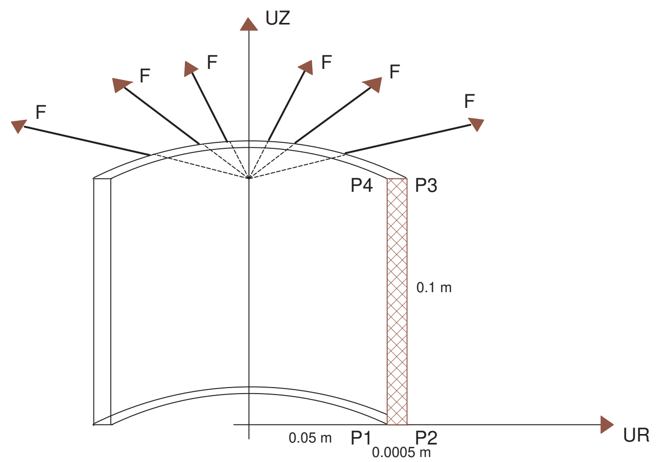
The structure is an axisymmetrical hollow cylinder embedded at the lower surface. It is subjected to imposed load at the upper surface: the radial strength outwards. The cylinder follows Ohno's viscoplastic law .

Goal Test the Ohno's model for solid structures.

Version 97' customer version

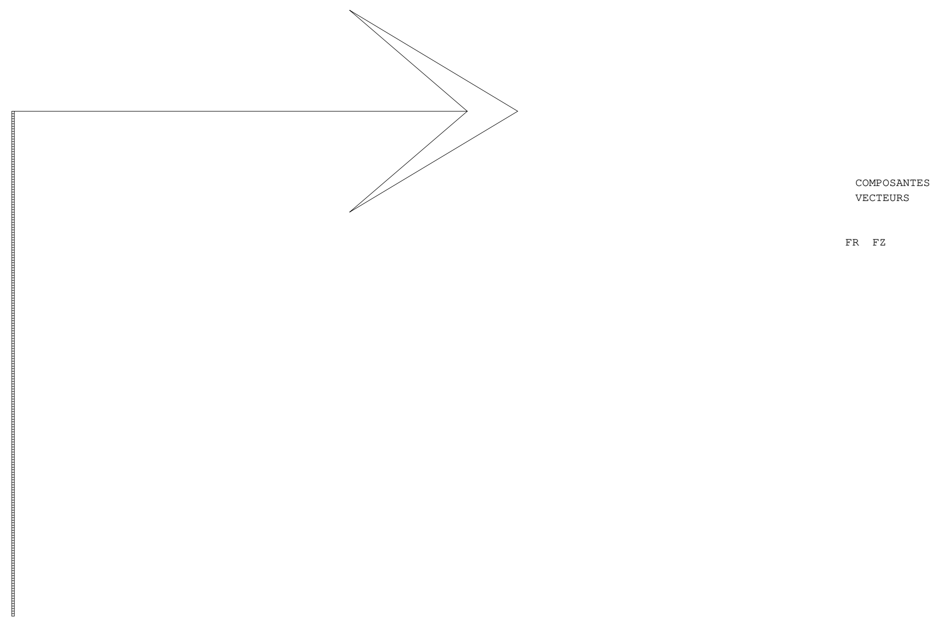
Model description

$$
\begin{array}{lcl}
E & = & 7.34\ 10^{10} \\
NU & = & 0.33 \\
RHO & = & 7.8\ 10^{3} \\
ALPHA & = & 1.\ 10^{-5}
\end{array}
$$



Test ohno2 **Results**

CASTEM FIGURES



APPLIED STRENGTH

FINAL STRAIN

```
*          Test Ohno2.dgibi: Jeux de données
*          ---------------------------------
*
*
**************************************************
*                                                *
*  TEST DE VALIDATION D'UNE LOI DE COMPORTEMENT  *
*  --------------------------------------------  *
*             DE MATERIAU                        *
*             -----------                        *
*                                                *
*  LOI DE COMPORTEMENT VISCOPLASTIQUE DE:        *
*   OHNO (CHABOCHE MODIFIE OHNO)                 *
*                                                *
*  TEST POUR DES ELEMENTS COQUE MINCE            *
*  ---------------------------------             *
*                                                *
*  MAILLAGES:                                    *
*   PREMIER MAILLAGE: ELEMENTS MASSIFS           *
*   DEUXIEME MAILLAGE: ELEMENTS COQUE MINCE      *
*   UN CYLINDRE CREUX AXISYMETRIQUE              *
*   LONGUEUR      L =.1 M                        *
*   RAYON INTERNE R =.05 M                       *
*   EPAISSEUR     E =.0005 M                     *
*                                                *
*  CHARGEMENT:                                   *
*   CYLINDRE ENCASTRE EN SON EXTREMITE BASSE     *
*   FORCES IMPOSEES SUR L'EXTREMITE DU HAUT:     *
*      FORCES RADIALES DIRIGEES VERS L'EXTERIEUR *
*                                                *
**************************************************
*
opti echo 0 dime 2 elem qua8 mode axis ;
*
*
*  Maillage
*
rayon0 = .05 ;
epais0 = .0005 ;
l1 = epais0 + rayon0 ;
l2 = .1 ;
n1 = 1 ;
n2 = 200 ;
p1 = rayon0 0. ;
p2 = l1 0. ;
p3 = l1 l2 ;
p4 = rayon0 l2 ;
d1 = d p1 n1 p2 ;
d2 = d p2 n2 p3 ;
d3 = d p3 n1 p4 ;
d4 = d p4 n2 p1 ;
vol1 = 'DALL' d1 d2 d3 d4 ;
*
*    Modele de calcul
*
mod0 = 'MODE' vol1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'VISCOPLASTIQUE' 'OHNO' ;
mat0 = 'MATE' mod0 'YOUN' 7.34E10 'NU' 0.33
       'RHO' 7.8E3 'ALPHA' 1.E-5 'N' 24
       'KK' 10.E6 'K0' 116.E6 'ALFK' 1.5 'ALFR' 0.35
       'ALF' 2.E6 'A1' 67.5E6 'C1' 1300 'BET1' 4807.E6
       'R1' 4 'A2' 80.E6 'C2' 45 'BET2' 58480.E6
       'R2' 4 'PHI' 1. 'B' 12 'GAMA' 2.E-7 'M' 2
       'QMAX' 455.E6 'QSTA' 200.E6 'MU' 19 'ETA' 0.06
       'EXP1' 2. 'EXP2' 2. ;
*
*    Conditions aux limites
*
```

```
* cl1 = 'BLOQ' d1 'UR' 'UZ' 'RT' 'RZ' ;
* cl0 = cl1 ;
* *
  *    Chargement
  *
  pres0 = 1.E4 ;
  p01 = ( rayon0 + ( epais0 / 2. ) ) l2 ;
  p00 = vol1 'POIN' 'PROC' p01 ;
  dep1 = 'FORCE' 'FR' pres0 p00 ;
  ev0 = 'EVOL' 'MANU' temps ( 'PROG' 0. 10000. ) y
        ( 'PROG' 0. 1. ) ;
  cha0 = 'CHAR' 'MECA' dep1 ev0 ;
  vec1 = 'VECT' dep1 1.E-5 'FR' 'FZ' 'ROUG' ;
  *
  *  Temps du calcul
  *
  dt0 = 20. ;
  tfin0 = 700. ;
  *
  *
  *  Resolution par PASAPAS
  *---------------------------------------
  *
  ta1 = 'TABLE' ;
  ta1 .'MODELE' = mod0 ;
  ta1 .'CARACTERISTIQUES' = mat0 ;
  ta1 .'BLOCAGES_MECANIQUES' = cl0 ;
  ta1 .'CHARGEMENT' = cha0 ;
  ta1 .'TEMPS_CALCULES' = 'PROG' 0. 'PAS' dt0 tfin0 ;
  *
  PASAPAS ta1 ;
  *
  *
  *  Post-traitement
  *-----------------------------------------
  *
  dim0 = 'DIME' ( ta1 .'TEMPS' ) ;
  char2 = 'PROG' 0. ;
  depl2 = 'PROG' 0. ;
  defi2 = 'PROG' 0. ;
  cont1 = 'PROG' 0. ;
  *
  reac0 = ta1 .'REACTIONS'.( dim0 - 1 ) ;
  depl0 = ta1 .'DEPLACEMENTS'.( dim0 - 1 ) ;
  vec2 = 'VECT' reac0 1.E-2 'FR' 'FZ' 'ROUG' ;
  defo0 = 'DEFO' vol1 depl0 0. 'ROUG' ;
  defo1 = 'DEFO' vol1 depl0 1. ;
  reac1 = 'REDU' reac0 p1 ;
  reac2 = 'REDU' reac0 p2 ;
  vec01 = 'VECT' reac1 1.E-2 'FR' 'FZ' 'ROUG' ;
  vec02 = 'VECT' reac2 1.E-2 'FR' 'FZ' 'VERT' ;
  *
  'REPETER' bloc0 ( dim0 - 1 ) ;
    i0 = &bloc0 ;
    reac0 = ta1 .'REACTIONS'.i0 ;
    depl0 = ta1 .'DEPLACEMENTS'.i0 ;
    sigm0 = ta1 .'CONTRAINTES'.i0 ;
    defi0 = ta1 .'VARIABLES_INTERNES'.i0 ;
    temp0 = ta1 .'TEMPS'.i0 ;
    depl1 = 'EXTR' depl0 'UR' p3 ;
    depl2 = depl2 'ET' ( 'PROG' depl1 ) ;
    defi1 = 'MAXI' ( 'EXCO' defi0 'EPSE' ) ;
    defi2 = defi2 'ET' ( 'PROG' defi1 ) ;
  *  cont0 =  'MAXI' ( 'EXCO' 'SMXX' sigm0 ) ;
  *  cont1 = cont1 'ET' ( 'PROG' cont0 ) ;
    char1 = pres0 * temp0 / tfin0 ;
    char2 = char2 'ET' ( 'PROG' char1 ) ;
  'FIN' bloc0 ;
```

```
*
opti echo 0 dime 2 elem seg2 mode axis ;
*
*
*  Maillage
*
epais0 = .0005 ;
rayon0 = .05 ;
l10 = .1 ;
n10 = 400 ;
p10 = rayon0 0. ;
p20 = rayon0 l10 ;
mail1 = d p10 n10 p20 ;
*
*    Modele de calcul
*
mod00 = 'MODE' mail1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'VISCOPLASTIQUE' 'OHNO' 'COQ2' ;
mat00 = 'MATE' mod00 'YOUN' 7.34E10 'NU' 0.33
       'RHO' 7.8E3 'N' 24
       'KK' 10.E6 'K0' 116.E6 'ALFK' 1.5 'ALFR' 0.35
       'ALF' 2.E6 'A1' 67.5E6 'C1' 1300 'BET1' 4807.E6
       'R1' 4 'A2' 80.E6 'C2' 45 'BET2' 58480.E6
       'R2' 4 'PHI' 1. 'B' 12 'GAMA' 2.E-7 'M' 2
       'QMAX' 455.E6 'QSTA' 200.E6 'MU' 19 'ETA' 0.06
       'EXP1' 2. 'EXP2' 2. ;
car00 = 'CARA' mod00 'EPAI' epais0 'ALFA' ( 2. / 3. ) ;
*
*    Conditions aux limites
*
cl10 = 'BLOQ' p10 'UR' 'UZ' 'RT' 'RZ' ;
cl00 = cl10 ;
*
*    Chargement
*
pres0 = 1.E4 ;
dep10 = 'FORCE' 'FR' pres0 p20 ;
ev00 = 'EVOL' 'MANU' temps ( 'PROG' 0. 10000. ) y
       ( 'PROG' 0. 1. ) ;
cha00 = 'CHAR' 'MECA' dep10 ev00 ;
vec1 = 'VECT' dep10 1.E-5 'FR' 'FZ' ;
*
*  Temps du calcul
*
dt00 = 20. ;
tfin00 = 700. ;
*
*
*  Resolution par PASAPAS
*---------------------------------------
*
ta10 = 'TABLE' ;
ta10 .'MODELE' = mod00 ;
ta10 .'CARACTERISTIQUES' = mat00 'ET' car00 ;
ta10 .'BLOCAGES_MECANIQUES' = cl00 ;
ta10 .'CHARGEMENT' = cha00 ;
ta10 .'TEMPS_CALCULES' = 'PROG' 0. 'PAS' dt00 tfin00 ;
*
PASAPAS ta10 ;
*
*
*  Post-traitement
*-----------------------------------------
*
dim10 = 'DIME' ta10 .'TEMPS' ;
depl12 = 'PROG' 0. ;
*
reac0 = ta10 .'REACTIONS'.( dim10 - 1 ) ;
depl0 = ta10 .'DEPLACEMENTS'.( dim10 - 1 ) ;
vec2 = 'VECT' reac0 1.E-2 'FR' 'FZ' 'ROUG' ;
defo0 = 'DEFO' mail1 depl0 0. ;
defo1 = 'DEFO' mail1 depl0 100. ;
*
'REPETER' bloc0 ( dim10 - 1 ) ;
  i10 = &bloc0 ;
  depl10 = ta10 .'DEPLACEMENTS'.i10 ;
  temp10 = ta10 .'TEMPS'.i10 ;
  depl11 = 'EXTR' depl10 'UR' p20 ;
  depl12 = depl12 'ET' ( 'PROG' depl11 ) ;
'FIN' bloc0 ;
*
*  Erreur
*
prob1 = faux ;
'REPETER' bloc0 ( dim10 - 1 ) ;
  i10 = &bloc0 ;
  depl1 = 'EXTR' depl2 i10 ;
  depl11 = 'EXTR' depl12 i10 ;
*
'SI' (depl1 'NEG' 0. ) ;
  err_e0 = 'ABS' ( ( depl1 - depl11 ) / depl1 ) ;
  err_e0 = err_e0 * 100. ;
'SINON' ;
  err_e0 = 0. ;
'FINSI' ;
*
*
  'SI' ( err_e0 '>' 3.E-1 ) ;
     prob0 = vrai ;
  'SINON' ;
     prob0 = faux ;
  'FINSI' ;
  'SI' (prob0 'EGA' vrai ) ;
     prob1 = vrai ;
  'FINSI' ;
*
'FIN' bloc0 ;
*
'SI' ( prob1 'EGA' vrai ) ;
   'ERRE' 5 ;
'SINON' ;
   'ERRE' 0 ;
'FINSI' ;
*
*
'FIN' ;
```

Test ohno2 **Comments**

- The Ohno viscoplastic model

```
MODO = MODE VOL1 MECANIQUE ELASTIQUE ISOTROPE
       VISCOPLASTIQUE OHNO ;
MATO = MATE MODO
    YOUN 7.34E10  NU 0.33
    RHO 7.8E3     ALPHA 1.E-5
    N 24          KK 10.E6     K0 116.E6
    ALFK 1.5      ALFR 0.35    ALF 2.E6
    A1 67.5E6     C1 1300      BET1 4807.E6  R1 4
    A2 80.E6      C2 45        BET2 58480.E6 R2 4
    PHI 1.        B 12         GAMA 2.E-7 M 2
    QMAX 455.E6   QSTA 200.E6  MU 19
    ETA 0.06      EXP1 2.      EXP2 2. ;
```

The Ohno model is close to the Chaboche viscoplastic one. The equations for this model are as follows:

-> *Criterion:* F = J2 (S-X) - RR - KK:

-> *Strain hardening:* dXi =2/3 * Ai*Ci*dEP - Ci*Xi*p'*< dEP:Ki >*(J2(Xi)/
(J2(Xi)/BETi)**(Ri-1) * Xi * dt (i=1 or 2)
if BETi is non zero, else:
dXi =2/3 * Ai*Ci*dEP - Ci*Xi*dp' (i=1 or 2)
with : p' = PHI+(1-PHI)* e** (-B*p)
LIMi = Ai/p' (i=1 ou 2)
Ki = Xi/J2(Xi)
dp = < F/K >**N * e**(ALF* < F/K >**(N+1))
dR = B*(Q - R)*dp + GAMA*Sign(QR-R)*|QR-R|**M *dt

-> *Flow rule:* dEP = 3/2 * dp * < S-X > / J2(S-X)

-> *Memory of inelastic strain* G = I2 (EP-Y) - q < 0
dQ = 2*MU*(QMAX-Q)* dq
dq = ETA * < n.nn > * dp
dY = (3/2 ** 0.5) * (1-ETA) * < n.nn > * nn*dp

QR = Q - QSTA * (1 - ((QMAX-Q)/QMAX)**2)

The initial value of Q is Q0 and must be initialized: Q = Q0 = 30 MPa (steel 316). For this purpose, a field of internal variables must be created, with a component named 'QQ' , its value being Q0. This field will be passed into the table of PASAPAS.

The parameter values for the steel 316L at 600C are given for information only. The data to be input are the following:

			Values for Steel 316
Threshold evolution law:			– – – – – – – – – –
'N '	:	viscoplasticity threshold exponent	24
'KK '	:	initial value for elastic limit	10 MPa
'K0 '	:	initial factor for the threshold normalization	116 MPa
'ALFK'	:	K isotropic evolution coefficient	1.5
'ALFR'	:	threshold isotropic evolution coefficient	0.35
'ALF '	:	viscosity coefficient	2.E6

			Values for Steel 316
Evolution law for the centres X1 and X2:			–––––––––––
'A1 '	:	plastic strain coefficient	67.5 MPa
'C1 '	:	reference term coefficient	1300
'BET1'	:	normalization factor for restoration	4807 MPa
'R1 '	:	restoration term exponent	4
'A2 '	:	plastic strain coefficient	80 MPa
'C2 '	:	reference term coefficient	45
'BET2'	:	normalization factor for restoration	5840 MPa
'R2 '	:	restoration term exponent	4
'PHI '	:	reference term multiplying coefficient	1.
Evolution law for isotropic stress hardening:			
'B '	:	isotropic stress hardening coefficient	12
'GAMA'	:	restoration effect coefficient	2.E-7
'M '	:	restoration term exponent	2
Evolution law for the plastic deformation memory:			
'QMAX'	:	Q maximal value	455 MPa
'QSTA'	:	Q stabilized value	200 MPa
'MU '	:	Q evolution law coefficient	19
'ETA '	:	factor linking q with the plastic strain	0.06

The following parameters will have to be specified with the elastic behavior.

–> *Notation:* S stress tensor
Xi kinematic stress hardening variables (i=1 ou 2)
EP inelastic strain tensor
p accumulated equivalent inelastic strain
q isotropic variable for the memory surface in deformation
Y kinematic variable for the memory surface in deformation
I2 tensor second invariant
J2 deviatoric tensor second invariant
n normal from F criterion
nn normal from G threshold
t time
$< a >$ positive part of a
x.y scalar product of x by y
X = X1 for a single centre
X1+X2 for two centres
RR = ALFR * R
K = K0 + ALFK * R
Q(0)=Q0

4.4 Test guionnet_tra Description sheet

Test name guionnet_tra

Calculation type ISOTROPIC VISCOPLASTIC MECHANICS **Finite element type** CUB8

Topic Guionnet model

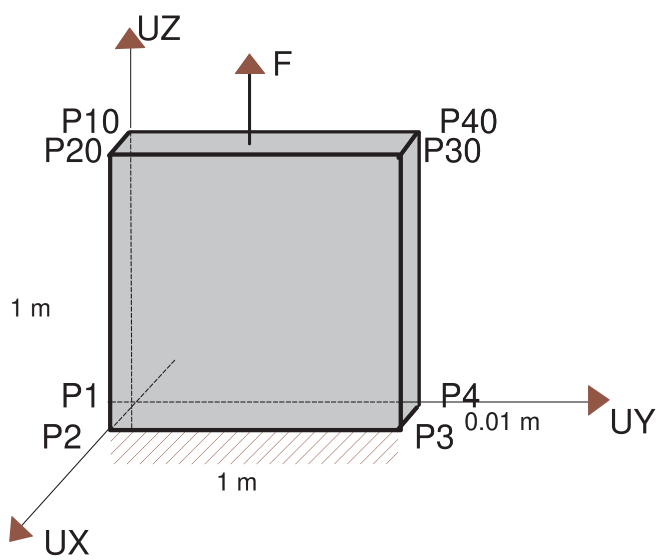
The structure is a square plate embedded at the lower surface and at the upper surface. It is subjected to imposed displacements at the upper surface: the tensile strength. The plate follows a viscoplastic law according the Guionnet's model.

Goal Test the Guionnet model for solid structures.

Version 97' customer version

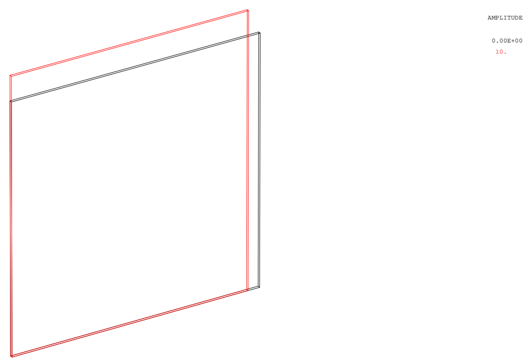
Model description

$$E = 192500$$
$$NU = 0.33$$

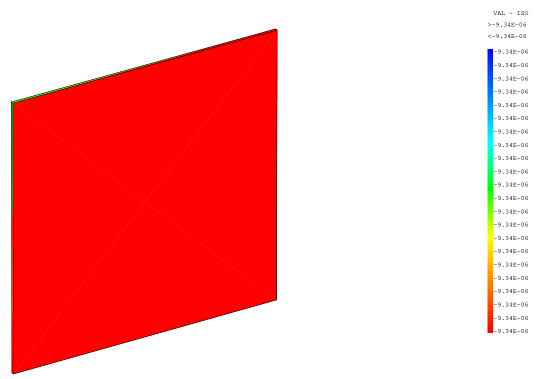


Test guionnet_tra Results

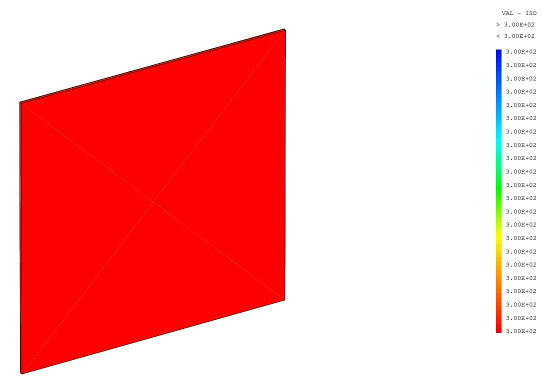
CASTEM FIGURES



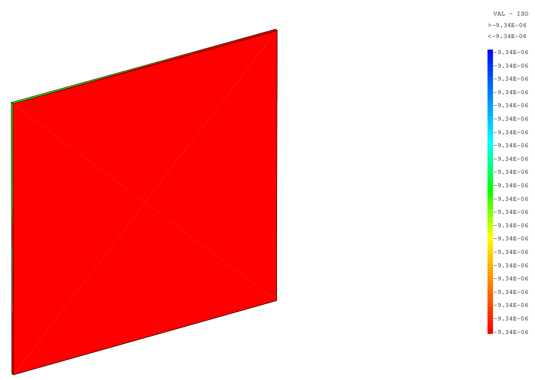
FINAL DEFORMATION



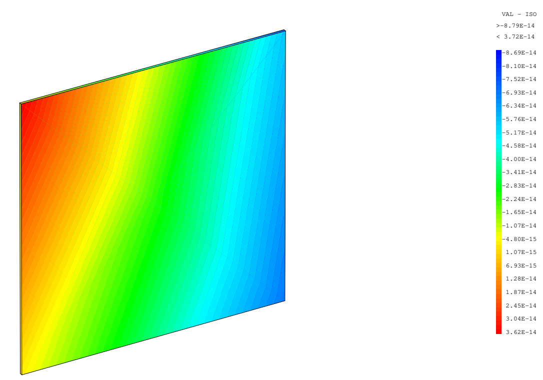
STRESS IN THE X DIRECTION



STRESS IN THE Z DIRECTION



STRESS IN THE Y DIRECTION



SHEAR STRESS IN THE YZ DIRECTION

```
*       Test Guionnet_tra.dgibi: Jeux de données
*       ---------------------------------------
*
**************************************************
*                                                *
*  TEST DE VALIDATION D'UNE LOI DE COMPORTEMENT  *
*  --------------------------------------------  *
*              DE MATERIAU                       *
*              -----------                       *
*                                                *
*  LOI DE COMPORTEMENT VISCOPLASTIQUE DE :       *
*            GUIONNET                            *
*                                                *
*  COMPARAISON DE CALCULS SUR DES ELEMENTS:      *
*     - COQUE EPAISSE ( MFR=5 )                  *
*     - MASSIF 3D     ( MFR=1 )                  *
*                                                *
*  STRUCTURE : UNE PLAQUE CARREE                 *
*     - COTE : 1. M                              *
*     - EPAISSEUR : 0.01  M                      *
*                                                *
*  CHARGEMENT : DEPLACEMENTS IMPOSES             *
*               ESSAI DE TRACTION                *
*                                                *
**************************************************
opti echo 0 dime 3 elem cub8 ;
*
*  Maillage
*
l1 = 1. ;
l2 = 1. ;
ep0 = .01 ;
p1 = 0. 0. 0. ;
p2 = ep0 0. 0. ;
p3 = ep0 l2 0. ;
p4 = 0. l2 0. ;
p10 = 0. 0. l1 ;
p20 = ep0 0. l1 ;
p30 = ep0 l2 l1 ;
p40 = 0. l2 l1 ;
li1 = d p1 1 p2 ;
li2 = d p2 1 p3 ;
li3 = d p3 1 p4 ;
li4 = d p4 1 p1 ;
li10 = d p10 1 p20 ;
li20 = d p20 1 p30 ;
li30 = d p30 1 p40 ;
li40 = d p40 1 p10 ;
sur1 = 'DALL' li1 li2 li3 li4 ;
sur10 = 'DALL' li10 li20 li30 li40 ;
mail1 = sur1 'VOLU' 1 sur10 ;
*
*  Modele
*
youn0 = 2.1E11 ;
nu0 = .3 ;
mod0 = 'MODE' mail1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'VISCOPLASTIQUE' 'GUIONNET' ;
mat0 = matr mod0 'YOUN' 192500.  'NU  ' 0.3   'N   ' 24
      'K   ' 200.    'CK  ' 0.       'R0  ' 25.
      'CD  ' 0.     'M   ' 0.8         'A   ' 24000.
      'A1  ' 3.3E5  'C   ' 180.   'C1  ' 40.
      'G   ' 0.     'R   ' 0.      'NN  ' 0.005
      'C2  ' 40.    'Q   ' 1.466    'G1  ' 0. 'R1  ' 0.
      'BETA' 1.1 'C0  ' 180.     'P1M0' 0.00333 ;

*
*  Conditions aux limites
* *
* cl1 = 'BLOQ' sur1 'UZ' ;
* cl2 = 'BLOQ' ( li1 'ET' li10 ) 'UY' ;
cl3 = 'BLOQ' ( li2 'ET' li20 ) 'UX' ;
cl4 = 'BLOQ' sur10 'UZ' ;
cl0 = cl1 'ET' cl2 'ET' cl3 'ET' cl4 ;
*
*  Chargement
*
depmax = .1 ;
dep0 = 'DEPI' cl4 depmax ;
ev0 = 'EVOL' 'MANU' temps ( 'PROG' 0. 1000. )
y ( 'PROG' 0. 1. ) ;
cha0 = 'CHAR' 'MECA' dep0 ev0 ;
*
dt0 = 10. ;
tfin0 = 100. ;
pro0 = 'PROG' 0. 'PAS' dt0 tfin0 ;
*
*  Appel à PASAPAS
*
ta1 = 'TABLE' ;
ta1 .'MODELE' = mod0 ;
ta1 .'CARACTERISTIQUES' = mat0 ;
ta1 .'CHARGEMENT' = cha0 ;
ta1 .'BLOCAGES_MECANIQUES' = cl0 ;
ta1 .'TEMPS_CALCULES' = pro0 ;
*
PASAPAS ta1 ;
*
*  Post traitement
*
dim0 = 'DIME' ta1 .'TEMPS' ;
*
*  Résultats calculés
*
reac0 = ta1 .'REACTIONS'.( dim0 - 1 ) ;
depl0 = ta1 .'DEPLACEMENTS'.( dim0 - 1 ) ;
temp0 = ta1 .'TEMPS'.( dim0 - 1 ) ;
sigm0 = ta1 .'CONTRAINTES'.(dim0 - 1 ) ;
eps0 = 'EPSI' mod0 depl0 ;
*
defo0 = 'DEFO' mail1 depl0 0. ;
defo1 = 'DEFO' mail1 depl0 10. 'ROUG' ;
*titr ' deformee finale' ;
*trace ( defo0 'ET' defo1 ) ;
*
sigm0 = 'CHAN' 'CHPO' sigm0 mod0 ;
*titr ' Contraintes suivant X' ;
*trace ( 'EXCO' sigm0 'SMXX' ) mail1 ;
*titr ' Contraintes suivant Y' ;
*trace ( 'EXCO' sigm0 'SMYY' ) mail1 ;
*titr ' Contraintes suivant Z' ;
*trace ( 'EXCO' sigm0 'SMZZ' ) mail1 ;
*titr ' Contraintes d cisaillement suivant YZ' ;
*trace ( 'EXCO' sigm0 'SMYZ' ) mail1 ;
reac0 = 'REDU' reac0 sur10 ;
reac0 = 'EXCO' reac0 'FZ' 'SCAL' ;
chp1 = 'MANU' 'CHPO' sur10 1 'SCAL' 1. ;
reac1 = 'XTY' reac0 chp1 ( 'MOTS' 'SCAL' )
        ( 'MOTS' 'SCAL' ) ;
*
opti echo 0 dime 3 elem qua8 ;
*
*  Maillage
*
p1 = 0. 0. 0. ;
p4 = 0. l2 0. ;
```

```
p10 = 0. 0. l1 ;
p40 = 0. l2 l1 ;
li1 = d p1 1 p4 ;
li2 = d p4 1 p40 ;
li3 = d p40 1 p10 ;
li4 = d p10 1 p1 ;
mail1 = 'DALL' li1 li2 li3 li4 ;
*titr ' Maillage' ;
*trace mail1 ;
*
*  Modele
*
mod0 = 'MODE' mail1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'VISCOPLASTIQUE' 'GUIONNET' 'COQ8' ;
mat0 = matr mod0 'YOUN' 192500.  'NU  ' 0.3
       'N   ' 24    'K   ' 200. 'CK  ' 0. 'A   ' 24000.
       'R0  ' 25.   'CD  ' 0.     'M   ' 0.8
       'A1  ' 3.3E5 'C   ' 180.   'C1  ' 40. 'G   ' 0.
       'R   ' 0.      'NN  ' 0.005   'C2  ' 40.
       'Q   ' 1.466  'G1  ' 0.    'R1  ' 0. 'BETA' 1.1
       'C0  ' 180.   'P1M0' 0.00333 ;
car0 = 'CARA' mod0 'EPAI' ep0 ;
*
*  Conditions aux limites
*
cl1 = 'BLOQ' li1 'UZ' ;
cl2 = 'BLOQ' li4 'UY' ;
cl3 = 'BLOQ' mail1 'UX' ;
cl4 = 'BLOQ' li3 'UZ' ;
cl0 = cl1 'ET' cl2 'ET' cl3 'ET' cl4 ;
*
*  Chargement
*
dep0 = 'DEPI' cl4 depmax ;
ev0 = 'EVOL' 'MANU' temps ( 'PROG' 0. 1000. )
y ( 'PROG' 0. 1. ) ;
cha0 = 'CHAR' 'DIMP' dep0 ev0 ;
*
*  Appel à PASAPAS
*
ta1 = 'TABLE' ;
ta1 .'MODELE' = mod0 ;
ta1 .'CARACTERISTIQUES' = mat0 'ET' car0 ;
ta1 .'CHARGEMENT' = cha0 ;
ta1 .'BLOCAGES_MECANIQUES' = cl0 ;
ta1 .'TEMPS_CALCULES' = pro0 ;
*
PASAPAS ta1 ;
*
*  Post traitement
*
dim0 = 'DIME' ta1 .'TEMPS' ;
*
*  Résultats calculés
*
reac0 = ta1 .'REACTIONS'.( dim0 - 1 ) ;
depl0 = ta1 .'DEPLACEMENTS'.( dim0 - 1 ) ;
temp0 = ta1 .'TEMPS'.( dim0 - 1 ) ;
sigm0 = ta1 .'CONTRAINTES'.( dim0 - 1 ) ;
eps0 = 'EPSI' mod0 depl0 car0 ;
*
defo0 = 'DEFO' mail1 depl0 0. ;
defo1 = 'DEFO' mail1 depl0 10. 'ROUG' ;
*titr ' deformee finale' ;
*trace ( defo0 'ET' defo1 ) ;
*
sigm0 = 'CHAN' 'CHPO' sigm0 mod0 ;
*titr ' Contraintes suivant Y' ;
*trace ( 'EXCO' sigm0 'SMSS' ) mail1 ;
*titr ' Contraintes suivant Z' ;
*trace ( 'EXCO' sigm0 'SMTT' ) mail1 ;
*titr ' Contraintes d cisaillement suivant YZ' ;
*trace ( 'EXCO' sigm0 'SMST' ) mail1 ;
reac0 = 'REDU' reac0 li3 ;
reac0 = 'EXCO' reac0 'FZ' 'SCAL' ;
chp1 = 'MANU' 'CHPO' li3 1 'SCAL' 1. ;
reac2 = 'XTY' reac0 chp1 ( 'MOTS' 'SCAL' )
        ( 'MOTS' 'SCAL' ) ;
*
'MESS' ' Réaction au sommet de la structure
, calcul massif        ='
reac1 ;
'MESS' ' Réaction au sommet de la structure
, calcul coque épaisse ='
reac2 ;
'SI' ( reac1 '>' 0. ) ;
   err1 = ( 'ABS' ( reac1 -reac2 ) ) / reac1 ;
   err1 = err1 * 100. ;
'SINON' ;
   err1 = 0. ;
'FINSI' ;
'MESS' ' Erreur obtenue en % =' err1 ;
*
'SI' ( err1 '>' 0.03 ) ;
   'ERRE' 5 ;
'SINON' ;
   'ERRE' 0 ;
'FINSI' ;
*
'FIN' ;
```

Test guionnet_tra **Comments**

- The Guionnet viscoplastic model

```
MODO = MODE MAIL1 MECANIQUE ELASTIQUE ISOTROPE
       VISCOPLASTIQUE GUIONNET ;
MATO = MATE MODO 'YOUN' 192500.    'NU  ' 0.3
      'N   ' 24      'K   ' 200    'CK  ' 0.
      'RO  ' 25.     'CD  ' 0.     'M   ' 0.8
      'A   ' 24000.  'A1  ' 3.3E5  'C   ' 180.
      'C1  ' 40.     'G   ' 0.     'R   ' 0.
      'NN  ' 0.005   'C2  ' 40.    'Q   ' 1.466
      'G1  ' 0.      'R1  ' 0.     'BETA' 1.1
      'C0  ' 180.    'P1M0' 0.00333 ;
```

The equations for the Guionnet model are as follows:

-> *Criterion:* F = J2 (S-X) - R - KK:

-> *Stress hardening:* dX = M* p1**(M-1) * [2/3 *(A*a2 + a1* EP.n)
- (C1 -C*a2)* X.n] *dEP
- (C0*< pI-P1M0 > + C*a4)*X*dp
- G * J2(X)**R * X*dt2/3 * Ai*Ci*dEP

and yield R = R0*(1-CD) + R0*CD*a2
KK = K *(1+CK*a2)
dEP = 3/2 * < F/KK >**N * n

if p1 > pI : da2 = da4 = NN * p1**(NN-1) * dp
if p1 < pI : da2 = C2*(Q * pI**NN -a2) * dp - G1 * a2**R1 *dt
if p1 < pI : da4 = - BETA*a4*dp/p

The parameter values for the steel 316L at 600C are given for information only. The data to be input are the following:

			Values for Steel 316
Threshold evolution law:			----------
'N '	:	viscoplasticity threshold exponent	20
'K '	:	initial factor for threshold normalization	10 MPa
'CK '	:	constant in K evolution law	3.87
'R0 '	:	initial value for the elastic limit	80 MPa
'CD '	:	constant in R evolution law	0.
Evolution law for the X centre:			
'A '	:	ALPHA2 coefficient	15000 MPa
'M '	:	plastic strain exponent (<1)	0.8
'A1 '	:	plastic strain coefficient	200000 MPa
'C '	:	ALPHA4 coefficient	40
'C1 '	:	reference term coefficient	180
'C0 '	:	tuning coefficient for ratchetting	0.
'P1M0'	:	threshold for tuning	
'G '	:	restoration term exponent	1.5E-10 /se
'R '	:	restoration term exponent	4.
Evolution law for the ALPHA2 and ALPHA4 internal variables			
'NN '	:	plastic strain exponent	0.075
'C2 '	:	plastic strain coefficient	4.
'Q '	:	plastic strain coefficient	3.43
'G1 '	:	restoration terme coefficient	1.5E-6
'R1 '	:	ALPHA2 exponent	4.
'BETA'	:	ALPHA2 coefficient	0.4

The following parameters will have to be specified with the elastic behavior.

–> *Notation:* S stress tensor
X kinematic stress hardening variable
ai internal variables (i=2, 4)
EP inelastic strain tensor
p accumulated equivalent inelastic strain
p1 dp1 = dp but p1=0 at each load reversal
pI value of p1 at load reversal
J2 second invariant of deviatoric tensor
n normal from F criterion
t time
$< a >$ positive part of a
x.y scalar product of x by y

For further details, refer to the CEA -N-2612 note.

4.5 Test zeril1 Description sheet

Test name zeril1

Calculation type ISOTROPIC ELASTOPLASTIC MECHANICS **Finite element type** CUB8

Topic Zerilli-Armstrong model

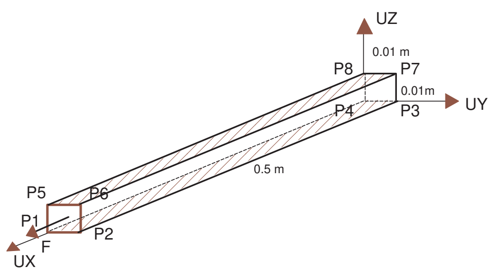
The structure is a beam embedded at the lower and at the upper surface. It is subjected to tensile strength (imposed displacements) at the end. The beam follows an elastoplastic law according to the Zerilli-Armstrong's model.

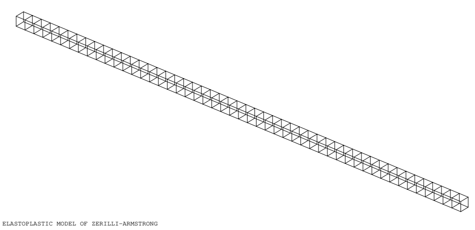
Goal Test the Zerilli-Armstrong model for BCC structure.

Version 97' customer version

Model description

E	=	$2.1\ 10^{11}$
NU	=	0.3
RHO	=	$7.8\ 10^{3}$
ALPHA	=	$1.\ 10^{-5}$





```
*          Test Zeril.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
**************************************************
*                                                *
*  TEST DE VALIDATION D'UNE LOI DE COMPORTEMENT  *
*  --------------------------------------------  *
*             DE MATERIAU                        *
*             -----------                        *
*                                                *
*  LOI DE COMPORTEMENT DE:                       *
*   ZERILLI-ARMSTRONG                            *
*   CAS CUBIQUE CENTRE ( C.C. )                  *
*                                                *
*  MAILLAGE:                                     *
*   UNE BARRE DE SECTION CARREE                  *
*   LONGUEUR L=.5 M                              *
*   LARGEUR  l=.01 M                             *
*                                                *
*  CHARGEMENT:                                   *
*   ESSAI DE TRACTION                            *
*   DEPLACEMENTS IMPOSES                         *
*                                                *
**************************************************
*
*
*
opti echo 0 dime 3 elem cub8 ;
*
*
*  Maillage
*
l1 = .5 ;
l2 = .01 ;
l3 = .01 ;
n1 = 50 ;
n2 = 1 ;
n3 = 1 ;
p1 = 0. 0. 0. ;
p2 = l1 0. 0. ;
p3 = l1 l2 0. ;
p4 = 0. l2 0. ;
p5 = 0. 0. l3 ;
p6 = l1 0. l3 ;
p7 = l1 l2 l3 ;
p8 = 0. l2 l3 ;
d1 = d p1 n3 p5 ;
d2 = d p5 n2 p8 ;
d3 = d p8 n3 p4 ;
d4 = d p4 n2 p1 ;
sur1 = 'DALL' d1 d2 d3 d4 ;
d5 = d p2 n3 p6 ;
d6 = d p6 n2 p7 ;
d7 = d p7 n3 p3 ;
d8 = d p3 n2 p2 ;
sur2 = 'DALL' d5 d6 d7 d8 ;
vol1 = sur1 'VOLU' n1 sur2 ;
*
*   Modele de calcul
*
mod0 = 'MODE' vol1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'PLASTIQUE' 'ZERILLI' ;
mat0 = 'MATE' mod0 'YOUN' 2.1E11 'NU' 0.3
       'RHO' 7.8E3 'ALPHA' 1.E-5 'DYG' 46.5E6
       'C1' 1033.E6 'C2' 890.E6 'C3' (300. * 698.E-5)
       'C4' ( 300. * 415.E-6 ) 'C5' 266.E6
       'N' 0.289 'K' 6.957E5 'L' 1E-3 'TYPE' 1. ;
*
```

```
*   Conditions aux limites
*
cl1 = 'BLOQ' sur1 'UX' ;
cl2 = 'BLOQ' sur2 'UX' ;
cl3 = 'BLOQ' p1 'UX' 'UY' 'UZ' ;
cl4 = 'BLOQ' p5 'UX' 'UY' ;
cl0 = cl1 'ET' cl2 'ET' cl3 'ET' cl4 ;
*
*   Chargement
*
dep1 = 'DEPI' cl2 .1 ;
ev0 = 'EVOL' 'MANU' temps ( 'PROG' 0. .01 ) y
      ( 'PROG' 0. 1. ) ;
cha0 = 'CHAR' 'DIMP' dep1 ev0 ;
*
*  Temps du calcul
*
dt0 = .00005 ;
tfin0 = .0005 ;
*
*
*  Resolution par PASAPAS
*--------------------------------------
*
ta1 = 'TABLE' ;
ta1 .'MODELE' = mod0 ;
ta1 .'CARACTERISTIQUES' = mat0 ;
ta1 .'BLOCAGES_MECANIQUES' = cl0 ;
ta1 .'CHARGEMENT' = cha0 ;
ta1 .'TEMPS_CALCULES' = 'PROG' 0. 'PAS' dt0 tfin0 ;
*
PASAPAS ta1 ;
*
*  Post-traitement
*----------------------------------------
*
dim0 = 'DIME' ta1 .'TEMPS' - 1 ;
YOUNG0 = 'EXCO' mat0 'YOUNG' 'SCAL' ;
YOUNG1 = 'MAXI' YOUNG0 ;
DYG0 = 'EXCO' mat0 'DYG' 'SCAL' ;
DYG1 = 'MAXI' DYG0 ;
C1 = 'EXCO' mat0 'C1' 'SCAL' ;
C11 = 'MAXI' C1 ;
C2 = 'EXCO' mat0 'C2' 'SCAL' ;
C21 = 'MAXI' C2 ;
C3 = 'EXCO' mat0 'C3' 'SCAL' ;
C31 = 'MAXI' C3 ;
C4 = 'EXCO' mat0 'C4' 'SCAL' ;
C41 = 'MAXI' C4 ;
C5 = 'EXCO' mat0 'C5' 'SCAL' ;
C51 = 'MAXI' C5 ;
XK0 = 'EXCO' mat0 'K' 'SCAL' ;
XK1 = 'MAXI' XK0 ;
XL0 = 'EXCO' mat0 'L' 'SCAL' ;
XL1 = 'MAXI' XL0 ;
XN0 = 'EXCO' mat0 'N' 'SCAL' ;
XN1 = 'MAXI' XN0 ;
TYPE0 = 'EXCO' mat0 'TYPE' 'SCAL' ;
TYPE1 = 'MAXI' TYPE0 ;
eps_el0 = ( DYG1 + ( XK1 * ( XL1 ** (-.5) ) ) ) /
          YOUNG1 ;
err_e1 = 0. ;
err_s1 = 0. ;
'REPETER' bloc1 dim0 ;
  i1 = &bloc1 ;
  depl1 = 'EXCO' ta1 .'DEPLACEMENTS'.i1 'UX' 'SCAL' ;
  depl0 = 'EXCO' ta1 .'DEPLACEMENTS'.(i1 - 1 ) 'UX'
          'SCAL' ;
```

```
  dep1 = 'MAXI' depl1 ;
  eps1 = dep1 / l1 ;
  dep0 = 'MAXI' depl0 ;
  sigm0 = ta1 .'CONTRAINTES'.i1 ;
  depeq0 = ta1 .'VARIABLES_INTERNES'.i1 ;
  def0 = ta1 .'DEFORMATIONS_INELASTIQUES'.i1 ;
  t1 = ta1 .'TEMPS'.i1 ;
  t0 = ta1 .'TEMPS'.(i1 - 1) ;
  dt0 = t1 - t0 ;
  dt0 = 0. ;
  'SI' ( ( 'ABS' dt0 ) < 1.E-10 ) ;
     deps0 = 0. ;
  'SINON' ;
     deps0 = ( dep1 - dep0 ) / ( dt0 * l1 ) ;
  'FINSI' ;
*
  sigx0 = 'MAXI' ( 'VMIS' mod0 sigm0 ) ;
  sigx00 = 'MINI' ( 'VMIS' mod0 sigm0 ) ;
  depx0 = 'MAXI' depeq0 ;
  depx00 = 'MINI' depeq0 ;
  sig_00 = 'EXCO' sigm0 'SMXX' 'SCAL' ;
  sig_0 = 'MAXI' sig_00 ;
  sig_y = 'MAXI' ( 'EXCO' sigm0 'SMYY' 'SCAL' ) ;
  sig_z = 'MAXI' ( 'EXCO' sigm0 'SMZZ' 'SCAL' ) ;
  def_00 = 'EXCO' def0 'EIXX' ;
  def_0 = 'MAXI' def_00 ;
*
*  Materiau cfc
*
  'SI' ( TYPE1 'EGA' 0. ) ;
*
     'SI' ( sigx0 '>EG'
     ( DYG1 + ( XK1 * ( XL1 ** (-.5) ) ) ) ) ;
        'SI' ( ( 'ABS' deps0 ) < 1.E-10 ) ;
           B0 = 0. ;
        'SINON' ;
           B0 = C21 * ( EXP ( ( C41 * ( LOG deps0 ) )
                - C31 ) ) ;
        'FINSI' ;
        A0 = YOUNG1 ;
        C0 = DYG1 + ( XK1 * ( XL1 ** (-.5) ) )
             - ( YOUNG1 * eps1 ) ;
*
        DELTA = ( B0 * B0 ) - ( 4. * A0 * C0 ) ;
        X1 = ( -1. * B0 - ( ( DELTA ) ** ( 0.5) ) ) /
             ( 2. * A0 ) ;
        X2 = ( -1. * B0 + ( ( DELTA ) ** ( 0.5) ) ) /
             ( 2. * A0 ) ;
        epsp0 = X2 * X2 ;
        sig0 = YOUNG1 * ( eps1 - epsp0 ) ;
*
     'SINON' ;
        epsp0 = 0. ;
        sig0 = sig_0 ;
     'FINSI' ;
*
  'SINON' ;
*
*  materiau CC
*
     LIM0 = DYG1 + ( XK1 * ( XL1 ** (-.5) ) ) ;
     'SI' ( ( 'ABS' deps0 ) < 1.E-10 ) ;
        LIM0 = LIM0 + 0. ;
     'SINON' ;
        LIM0 = LIM0 +
     ( C11 * ( EXP ( ( C41 * ( LOG deps0 ) )
      - C31 ) ) ) ;
     'FINSI' ;
     'SI' ( sig_0 > LIM0 ) ;
        A0 = YOUNG1 ;
        'SI' ( ( 'ABS' deps0 ) < 1.E-10 ) ;
           B0 = 0. ;
        'SINON' ;
           B0 = C11 * ( EXP ( ( C41 * ( LOG deps0 ) )
                - C31 ) ) ;
        'FINSI' ;
        B0 = B0 + DYG1 + ( XK1 * ( XL1 ** (-.5) ) )
             - ( YOUNG1 * eps1 ) ;
        C0 = C51 ;
*
        x0 = 0. ;
        x2 = x0 ;
        y2 = A0 * x2 ;
        y2 = y2 + ( C0 * ( x2 ** XN1 ) ) ;
        y2 = y2 + B0 ;
*
        x1 = eps1 ;
        x2 = x1 ;
        y2 = A0 * x2 ;
        y2 = y2 + ( C0 * ( x2 ** XN1 ) ) ;
        y2 = y2 + B0 ;
*
       'REPETER' bloc2 ;
*
          x2 = ( x0 + x1 ) / 2. ;
          y2 = A0 * x2 ;
          y2 = y2 + ( C0 * ( x2 ** XN1 ) ) ;
          y2 = y2 + B0 ;
*
          'SI' ( y2 > 1.E-5 ) ;
              x1 = x2 ;
          'SINON' ;
              'SI' ( y2 <  -1.E-5 ) ;
                  x0 = x2 ;
              'SINON' ;
                  epsp0 = x2 ;
                  sig0 = YOUNG1 * ( eps1 - x2 ) ;
                  'QUITTER' bloc2 ;
              'FINSI' ;
          'FINSI' ;
*
       'FIN' bloc2 ;
*
     'SINON' ;
        epsp0 = 0. ;
        sig0 = sig_0 ;
     'FINSI' ;
*
  'FINSI' ;
*
  err_s0 = 'ABS' ( sig0 - sig_0 ) ;
  err_s0 = err_s0 / sig0 ;
  err_s0 = err_s0 * 100. ;
  err_e0 = 'ABS' ( epsp0 -  def_0 ) ;
  'SI' (def_0 '>' ( 1.E-1 * eps_el0 ) ) ;
     err_e0 = err_e0 / def_0 ;
  'SINON' ;
     err_e0 = 0. ;
  'FINSI' ;
  err_e0 = err_e0 * 100. ;
*
*  Erreur trop grande ?
*
  'SI' ( err_e0 > 1. ) ;
     'ERRE' 5 ;
     err_e1 = 100. ;
```

```
  'SINON' ;
     err_e1 = err_e1 + 0. ;
  'FINSI' ;
  'SI' ( err_s0 > 1. ) ;
     'ERRE' 5 ;
     err_s1 = 100. ;
  'SINON' ;
     err_s1 = err_s1 + 0. ;
  'FINSI' ;
*
'FIN' bloc1 ;
*
'SI' ( ( err_e1 < 1. ) 'ET' ( err_s1 < 1. ) ) ;
   'ERRE' 0 ;
'FINSI' ;
*
*
'FIN' ;
```

Test zeril1 **Comments**

- The Zerilli-Armstrong elastoplastic model

```
MODO = MODE VOL1 MECANIQUE ELASTIQUE ISOTROPE
       PLASTIQUE ZERILLI ;
MATO = MATE MODO YOUN 2.1E11 NU 0.3
       RHO 7.8E3 ALPHA 1.E-5 DYG 46.5E6
       C1 1033.E6 C2 890.E6 C3 (300. * 698.E-5)
       C4 ( 300. * 415.E-6 ) C5 266.E6
       N 0.289 K 6.957E5 L 1E-3 TYPE 1. ;
```

The Zerilli-Armstrong model proposes a mathematical formulation of the Von Mises's yielding stress based on the theory of dislocations. The constitutive equations of this model are as follow:
-Yielding limit Y for Face Centered Cubic materials (F.C.C.):
Y = DYG+C2'.sqrt(P').exp(-c3'.T+C4'.T.ln(EPT))+K.L**(-1/2)
-Yielding limit Y for Body Centered Cubic materials (B.C.C.):
Y = DYG+C1'.exp(-C3'.T+C4'.T.ln(EPT))+C5'.(P')**N+K.L**(-.5)
with:
T: temperature
P': equivalent plastic strain: P'=sqrt(2/3.EP:EP)
EP: plastic strains
EPT: equivalent strain rate: EPT=sqrt(2/3.ET:ET)
ET: strain rate
The parameters to be input in the Zerilli-Armstrong model are DYG, Ci, N, K, L as well as TYPE:

'DYG'	: DYG term
'C1'	: C1' coefficient
'C2'	: C2' coefficient
'C3'	: C3'.T term (C3' multiplicated by T) entered as an EVOLUTION object (the parameter is the temperature)
'C4'	: C4'.T term (C4' multiplicated by T) entered as an EVOLUTION object (the parameter is the temperature)
'C5'	: C5' coefficient
'N'	: N coefficient
'K'	: K coefficient
'L'	: average grain diameter
'TYPE'	: material structure if the material is FCC: TYPE=0. if the material is BCC: TYPE=1.

4.6 Test preston1 Description sheet

Test name preston1

Calculation type ISOTROPIC ELASTOPLASTIC MECHANICS **Finite element type** CUB8

Topic Preston Tonks Wallace model

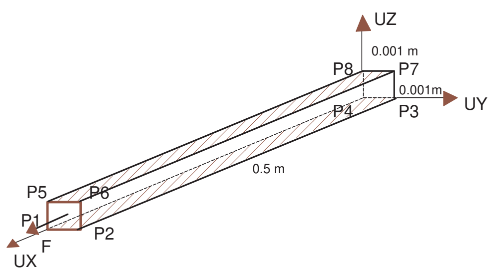
The structure is a beam embedded at the lower and upper surface. It is subjected to tensile strength (imposed displacements) at the end. The beam follows an elastoplastic law according to the Preston-Tonks-Wallace model.

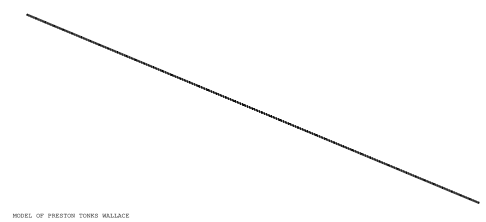
Goal Test the Preston Tonks Wallace model for BCC 3 structure.

Version 97' customer release

Model description

$$
\begin{aligned}
E &= 2.1\ 10^{11} \\
NU &= 0.3 \\
RHO &= 7.8\ 10^{3} \\
TAU &= 0.025
\end{aligned}
$$





```
*       Test Preston1.dgibi: Jeux de données
*       -----------------------------------
*
complete = faux;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
*************************************************
**************************************************
*                                                *
*  TEST DE VALIDATION D'UNE LOI DE COMPORTEMENT  *
*  --------------------------------------------  *
*             DE MATERIAU                        *
*             -----------                        *
*                                                *
*  LOI DE COMPORTEMENT DE:                       *
*   PRESTON TONKS WALLACE                        *
*   CAS CUBIQUE CENTRE ( C.C. )                  *
*                                                *
*  MAILLAGE:                                     *
*   UNE BARRE DE SECTION CARREE                  *
*   LONGUEUR L=.5 M                              *
*   LARGEUR  l=.001 M                            *
*                                                *
*  CHARGEMENT:                                   *
*   ESSAI DE TRACTION                            *
*   DEPLACEMENTS IMPOSES                         *
*                                                *
**************************************************
*
opti echo 0 dime 3 elem cub8 ;
*
*
*  Maillage
*
l1 = .5 ;
l2 = .001 ;
l3 = .001 ;
n1 = 50 ;
n2 = 1 ;
n3 = 1 ;
p1 = 0. 0. 0. ;
p2 = l1 0. 0. ;
p3 = l1 l2 0. ;
p4 = 0. l2 0. ;
p5 = 0. 0. l3 ;
p6 = l1 0. l3 ;
p7 = l1 l2 l3 ;
p8 = 0. l2 l3 ;
d1 = d p1 n3 p5 ;
d2 = d p5 n2 p8 ;
d3 = d p8 n3 p4 ;
d4 = d p4 n2 p1 ;
sur1 = 'DALL' d1 d2 d3 d4 ;
d5 = d p2 n3 p6 ;
d6 = d p6 n2 p7 ;
d7 = d p7 n3 p3 ;
d8 = d p3 n2 p2 ;
sur2 = 'DALL' d5 d6 d7 d8 ;
vol1 = sur1 'VOLU' n1 sur2 ;
*
*    Modele de calcul
*
mod0 = 'MODE' vol1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'PLASTIQUE' 'PRESTON' ;
mat0 = 'MATE' mod0 'YOUN' 2.1E11 'NU' 0.3
       'RHO' 7.8E3 'TAU' 0.025
       'P' 0. 'S0' 0.0085 'SINF' 0.00055
*          'K' 0. 'G' 0.00001
*          'Y0' 0.0001 'YINF' 0.0001 'Y1' 0.094 'Y2' 0.575
*          'BETA' 0.25 'GP' 3.00 'GT' 0. 'MU' 1.97
           'TMO' 1220. ;
*
*    Conditions aux limites
*
cl1 = 'BLOQ' sur1 'UX' ;
cl2 = 'BLOQ' sur2 'UX' ;
cl3 = 'BLOQ' p1 'UX' 'UY' 'UZ' ;
cl4 = 'BLOQ' p5 'UX' 'UY' ;
cl0 = cl1 'ET' cl2 'ET' cl3 'ET' cl4 ;
*
*    Chargement
*
dep1 = 'DEPI' cl2 .1 ;
ev0 = 'EVOL' 'MANU' temps ( 'PROG' 0. 5.E-4 ) y
      ( 'PROG' 0. 1. ) ;
cha0 = 'CHAR' 'DIMP' dep1 ev0 ;
*
*  Temps du calcul
*
dt0 = 1.E-5 ;
si complet;
tfin0 = 5.E-4 ;
sinon;
tfin0 = 4.e-5;
finsi;
*
*
*  Resolution par PASAPAS
*--------------------------------------
*
ta1 = 'TABLE' ;
ta1 .'MODELE' = mod0 ;
ta1 .'CARACTERISTIQUES' = mat0 ;
ta1 .'BLOCAGES_MECANIQUES' = cl0 ;
ta1 .'CHARGEMENT' = cha0 ;
*ta1 .'PRECISION' = 1.E-8 ;
ta1 .'TEMPS_CALCULES' = 'PROG' 0. 'PAS' dt0 tfin0 ;
*
PASAPAS ta1 ;
*
*
*  Post-traitement
*----------------------------------------
*
sig_3dy = 'PROG' 0. ;
sig_3dz = 'PROG' 0. ;
sigx1 = 'PROG' 0. ;
sigx2 = 'PROG' 0. ;
depx1 = 'PROG' 0. ;
depx2 = 'PROG' 0. ;
sig_1 = 'PROG' 0. ;
def_1 = 'PROG' 0. ;
epsp1 = 'PROG' 0. ;
err1 = 'PROG' 0. ;
err2 = 'PROG' 0. ;
sig1 = 'PROG' 0. ;
temp1 = 'PROG' 0. ;
epst1 = 'PROG' 0. ;
dim0 = 'DIME' ta1 .'TEMPS' - 1 ;
mod0 = ta1 .'MODELE' ;
YOUNG0 = 'EXCO' mat0 'YOUNG' 'SCAL' ;
YOUNG1 = 'MAXI' YOUNG0 ;
XNU0 = 'EXCO' mat0 'NU' 'SCAL' ;
XNU1 = 'MAXI' XNU0 ;
RHO0 = 'MAXI' ( 'EXCO' mat0 'RHO' 'SCAL' ) ;
```

```
TAU0 = 'EXCO' mat0 'TAU' 'SCAL' ;
TAU1 = 'MAXI' TAU0 ;
P1 = 'EXCO' mat0 'P' 'SCAL' ;
P11 = 'MAXI' P1 ;
S0 = 'EXCO' mat0 'S0' 'SCAL' ;
S01 = 'MAXI' S0 ;
SINF0 = 'EXCO' mat0 'SINF' 'SCAL' ;
SINF1 = 'MAXI' SINF0 ;
K0 = 'EXCO' mat0 'K' 'SCAL' ;
K1 = 'MAXI' K0 ;
G0 = 'EXCO' mat0 'G' 'SCAL' ;
G1 = 'MAXI' G0 ;
Y00 = 'EXCO' mat0 'Y0' 'SCAL' ;
Y01 = 'MAXI' Y00 ;
YINF0 = 'EXCO' mat0 'YINF' 'SCAL' ;
YINF1 = 'MAXI' YINF0 ;
Y10 = 'EXCO' mat0 'Y1' 'SCAL' ;
Y11 = 'MAXI' Y10 ;
Y20 = 'EXCO' mat0 'Y2' 'SCAL' ;
Y21 = 'MAXI' Y20 ;
BETA0 = 'EXCO' mat0 'BETA' 'SCAL' ;
BETA1 = 'MAXI' BETA0 ;
GP0 = 'EXTR' ( 'EXCO' mat0 'GP' 'SCAL' ) 'SCAL' 1 1 1 ;
GT0 = 'EXTR' ( 'EXCO' mat0 'GT' 'SCAL' ) 'SCAL' 1 1 1 ;
A1 = 2. * ( 1. + XNU1 ) ;
B1 = A1 / ( 3. * ( 1. - ( 2. * XNU1 ) ) ) ;
'REPETER' bloc1 dim0 ;
  i1 = &bloc1 ;
  depl0 = ta1 .'DEPLACEMENTS'.i1 ;
  temp0 = ta1 .'TEMPS'.i1 ;
  temp1 = temp1 'ET' ( 'PROG' temp0 ) ;
  dex1 = 'EXCO' depl0 'UX' 'SCAL' ;
  dex0 = 'EXCO' ta1 .'DEPLACEMENTS'.( i1 - 1 )
         'UX' 'SCAL' ;
  reac0 = ta1 .'REACTIONS'.i1 ;
  reac1 = 'MAXI' ( 'EXCO' reac0 'FX' 'SCAL' ) ;
  dep0 = 'MAXI' dex0 ;
  dep1 = 'MAXI' dex1 ;
  eps1 = 'EPSI' mod0 depl0 ;
  epsx1 = 'EXCO' eps1 'EPXX' 'SCAL' ;
  epsy1 = 'EXCO' eps1 'EPYY' 'SCAL' ;
  epsz1 = 'EXCO' eps1 'EPZZ' 'SCAL' ;
  treps0 = 'MAXI' ( epsx1 + epsy1 + epsz1 ) ;
  epsx10 = 'MAXI' epsx1 ;
  sigm0 = ta1 .'CONTRAINTES'.i1 ;
  depeq0 = ta1 .'VARIABLES_INTERNES'.i1 ;
  def0 = ta1 .'DEFORMATIONS_INELASTIQUES'.i1 ;
  t1 = ta1 .'TEMPS'.i1 ;
  t0 = ta1 .'TEMPS'.(i1 - 1) ;
  dt0 = t1 - t0 ;
  dt0 = 0. ;
  'SI' ( ( 'ABS' dt0 ) < 1.E-10 ) ;
     deps0 = 0. ;
  'SINON' ;
     deps0 = ( dep1 - dep0 ) / ( dt0 * l1 ) ;
  'FINSI' ;
*
  sigx0 = 'MAXI' ( 'VMIS' mod0 sigm0 ) ;
  sigx00 = 'MINI' ( 'VMIS' mod0 sigm0 ) ;
  depx0 = 'MAXI' ( 'EXCO' depeq0 'EPSE' 'SCAL' ) ;
  depx00 = 'MINI' ( 'EXCO' depeq0 'EPSE' 'SCAL' ) ;
  sig_00 = 'EXCO' sigm0 'SMXX' 'SCAL' ;
  sig_0 = 'MAXI' sig_00 ;
  eps_el0 = sig_0 / YOUNG1 ;
  sig_y = 'MAXI' ( 'EXCO' sigm0 'SMYY' 'SCAL' ) ;
  sig_z = 'MAXI' ( 'EXCO' sigm0 'SMZZ' 'SCAL' ) ;
  def_00 = 'EXCO' def0 'EIXX' ;
  def_0 = 'MAXI' def_00 ;
  def1 = 'MAXI' ( 'EXCO' def0 'EIXX' ) ;
  def2 = 'MAXI' ( 'EXCO' def0 'EIYY' ) ;
  def3 = 'MAXI' ( 'EXCO' def0 'EIZZ' ) ;
  def4 = 'MAXI' ( 'EXCO' def0 'GIXY' ) ;
  def5 = 'MAXI' ( 'EXCO' def0 'GIXZ' ) ;
  def6 = 'MAXI' ( 'EXCO' def0 'GIYZ' ) ;
  defp1 = ( def1 * def1 ) + ( def2 * def2 ) +
          ( def3 * def3 ) ;
  defp1 = defp1 - ( def1 * def2 ) - ( def2 * def3 ) -
          ( def1 * def3 ) ;
  defp2 = ( def4 * def4 ) + ( def5 * def5 ) +
          ( def6 * def6 ) ;
  defp0 = defp1 + ( 3. * defp2 ) ;
  defp0 = ( defp0 ** .5 ) * 2. / 3. ;
  sigx1 = sigx1 'ET' ( 'PROG' sigx0 ) ;
  sigx2 = sigx2 'ET' ( 'PROG' sigx00 ) ;
  depx1 = depx1 'ET' ( 'PROG' depx0 ) ;
  depx2 = depx2 'ET' ( 'PROG' depx00 ) ;
  sig_1 = sig_1 'ET' ( 'PROG' sig_0 ) ;
  def_1 = def_1 'ET' ( 'PROG' def_0 ) ;
  sig_3dy = sig_3dy 'ET' ( 'PROG' ( sig_y / sig_0 ) ) ;
  sig_3dz = sig_3dz 'ET' ( 'PROG' ( sig_y / sig_0 ) ) ;
  epst1 = epst1 'ET' ( 'PROG' epsx10 ) ;
*
*  Densite RHO1
*
  RHO1 = RHO0 / ( 1. + treps0 ) ;
  eta = RHO1 / RHO0 ;
*
*  Module de cisaillement G00
*
  G00 = YOUNG1 / ( 2. * ( 1. + XNU1 ) ) ;
*
*  Pulsation de Debye W0
*
  W0 = ( G00 / RHO1 ) ** ( .5 ) ;
  DTETA0 = ( 4. / 3.1416 ) ** ( .5 ) ;
  DTETA0 = DTETA0 * W0 / 6. ;
  DTETA0 = deps0 / DTETA0 ;
*
*  Contrainte de saturation
*
  'SI' ( ( 'ABS' dt0 ) < 1.E-10 ) ;
     YS1 = SINF1 ;
  'SINON' ;
     YS1 = 'ERF' ( K1 * ( 'LOG' ( G1 / DTETA0 ) ) ) ;
     YS1 = S01 - ( ( S01 - SINF1 ) * YS1 ) ;
  'FINSI' ;
  YS2 = S01 * ( ( DTETA0 / G1 ) ** ( BETA1 ) ) ;
  YS0 = ( 'ABS' ( YS1 - YS2 ) ) + YS1 + YS2 ;
  YS0 = YS0 / 2. ;
*
*  Limite d'élasticité
*
  'SI' ( ( 'ABS' dt0 ) < 1.E-10 ) ;
     YL1 = YINF1 ;
  'SINON' ;
     YL1 = 'ERF' ( K1 * ( 'LOG' ( G1 / DTETA0 ) ) ) ;
     YL1 = Y01 - ( ( Y01 - YINF1 ) * YL1 ) ;
  'FINSI' ;
  YL2 = Y11 * ( ( DTETA0 / G1 ) ** ( Y21 ) ) ;
  YL3 = ( 'ABS' ( YL2 - YS2 ) ) - YL2 - YS2 ;
  YL3 = -1. * YL3 / 2. ;
  YL0 = ( 'ABS' ( YL1 - YL3 ) ) + YL1 + YL3 ;
  YL0 = YL0 / 2. ;
*
  'SI' ( sig_0 '>' ( YL0 * G00 ) ) ;
*
```

```
*  Materiau C.C.
*
  'SI' ( P11 'EGA' 0. ) ;
*
        x0 = 0. ;
        x2 = x0 ;
        tr1 = treps0 - ( A1 * ( epsx10 - x2 ) /
              ( 3. * B1 ) ) ;
        tr1 = tr1 / 3. ;
        y2 = 'EXP' ( -1. * TAU1 * ( x2 - tr1 ) /
             ( YS0 - YL0 ) ) ;
        y2 = YS0 - ( ( YS0 - YL0 ) * y2 ) ;
        y2 = y2 * G00 ;
        y2 = y2 + ( YOUNG1 * x2 ) ;
        y2 = y2 - ( YOUNG1 * epsx10 ) ;
*
        x1 = epsx10 ;
        x2 = x1 ;
        tr1 = treps0 - ( A1 * ( epsx10 - x2 ) /
              ( 3. * B1 ) ) ;
        tr1 = tr1 / 3. ;
        y2 = 'EXP' ( -1. * TAU1 * ( x2 - tr1 ) /
             ( YS0 - YL0 ) ) ;
        y2 = YS0 - ( ( YS0 - YL0 ) * y2 ) ;
        y2 = y2 * G00 ;
        y2 = y2 + ( YOUNG1 * x2 ) ;
        y2 = y2 - ( YOUNG1 * epsx10 ) ;
*
        'REPETER' bloc2 ;
        x2 = ( x1 + x0 ) / 2. ;
        tr1 = treps0 - ( A1 * ( epsx10 - x2 ) /
              ( 3. * B1 ) ) ;
        tr1 = tr1 / 3. ;
        y2 = 'EXP' ( -1. * TAU1 * ( x2 - tr1 ) /
             ( YS0 - YL0 ) ) ;
        y2 = YS0 - ( ( YS0 - YL0 ) * y2 ) ;
        y2 = y2 * G00 ;
        y2 = y2 + ( YOUNG1 * x2 ) ;
        y2 = y2 - ( YOUNG1 * epsx10 ) ;
*
          'SI' ( y2 > 1.E-5 ) ;
              x1 = x2 ;
          'SINON' ;
            'SI' ( y2 <  -1.E-5 ) ;
              x0 = x2 ;
            'SINON' ;
              epsp0 = x2 ;
              epsp1 = epsp1 'ET' ( 'PROG' epsp0 ) ;
              den0 = A1 * GP0 * ( epsx10 - x2 ) / 3. ;
              den0 = den0 / ( eta ** ( 1. / 3. ) ) ;
              sup0 = A1 * ( G00 + GT0 ) *
                     ( epsx10 - x2 ) ;
              sig0 = sup0 / ( 1. + den0 ) ;
              sig1 = sig1 'ET' ( 'PROG' sig0 ) ;
              'QUITTER' bloc2 ;
            'FINSI' ;
          'FINSI' ;
       'FIN' bloc2 ;
*
  'SINON' ;
*
*  autres materiaux
*
       x0 = 0. ;
       x2 = x0 ;
       ex0 = P11 * ( YS0 - YL0 ) / ( S01 - YL0 ) ;
       ex1 = ( S01 - YL0 ) * ( ( 'EXP' ex0 ) - 1. ) ;
       tr1 = treps0 - ( A1 * ( epsx10 - x2 ) /
             ( 3. * B1 ) ) ;
       tr1 = tr1 / 3. ;
       y2 = 'EXP' ( -1. * P11 * TAU1 *
            ( x2 - tr1 ) / ex1 ) ;
       y2 = ( 1. -  ( 'EXP' ( -1. * ex0 ) ) ) * y2 ;
       y2 = 'LOG' ( 1. - y2 ) ;
       y2 = ( ( S01 - YL0 ) * y2 / P11 ) + YS0 ;
       y2 = y2 * G00 ;
       y2 = y2 + ( YOUNG1 * x2 ) ;
       y2 = y2 - ( YOUNG1 * epsx10 ) ;
       y0 = y2 ;
*
       x1 = epsx10 ;
       x2 = x1 ;
       tr1 = treps0 - ( A1 * ( epsx10 - x2 ) /
             ( 3. * B1 ) ) ;
       tr1 = tr1 / 3. ;
       y2 = 'EXP' ( -1. * P11 * TAU1 *
            ( x2 - tr1 ) / ex1 ) ;
       y2 = ( 1. -  ( 'EXP' ( -1. * ex0 ) ) ) * y2 ;
       y2 = 'LOG' ( 1. - y2 ) ;
       y2 = ( ( S01 - YL0 ) * y2 / P11 ) + YS0 ;
       y2 = y2 * G00 ;
       y2 = y2 + ( YOUNG1 * x2 ) ;
       y2 = y2 - ( YOUNG1 * epsx10 ) ;
       y1 = y2 ;
*
       'REPETER' bloc2 ;
       x2 = ( x0 + x1 ) / 2. ;
       tr1 = treps0 - ( A1 * ( epsx10 - x2 ) /
             ( 3. * B1 ) ) ;
       tr1 = tr1 / 3. ;
       y2 = 'EXP' ( -1. * P11 * TAU1 *
            ( x2 - tr1 ) / ex1 ) ;
       y2 = ( 1. -  ( 'EXP' ( -1. *  ex0 ) ) ) * y2 ;
       y2 = 'LOG' ( 1. - y2 ) ;
       y2 = ( ( S01 - YL0 ) * y2 / P11 ) + YS0 ;
       y2 = y2 * G00 ;
       y2 = y2 + ( YOUNG1 * x2 ) ;
       y2 = y2 - ( YOUNG1 * epsx10 ) ;
*
          'SI' ( y2 > ( 1.E-8 * ( 'ABS' ( y1 - y0 ))));
              x1 = x2 ;
          'SINON' ;
            'SI' (y2 < ( -1.E-8 * ('ABS'( y0 - y1 ))));
              x0 = x2 ;
            'SINON' ;
              epsp0 = x2 ;
              epsp1 = epsp1 'ET' ( 'PROG' epsp0 ) ;
              den0 = A1 * GP0 * ( epsx10 - x2 ) / 3. ;
              den0 = den0 / ( eta ** ( 1. / 3. ) ) ;
              sup0 = A1 * ( G00 + GT0 ) *
                     ( epsx10 - x2 ) ;
              sig0 = sup0 / ( 1. + den0 ) ;
              sig1 = sig1 'ET' ( 'PROG' sig0 ) ;
              'QUITTER' bloc2 ;
            'FINSI' ;
          'FINSI' ;
       'FIN' bloc2 ;
  'FINSI' ;
*
  'SINON' ;
       epsp0 = 0. ;
       epsp1 = epsp1 'ET' ( 'PROG' epsp0 ) ;
       sig0 = sig_0 ;
       sig1 = sig1 'ET' ( 'PROG' sig0 ) ;
  'FINSI' ;
*
```

```
  err_s0 = 'ABS' ( sig0 - sig_0 ) ;
  err_s0 = err_s0 / sig0 ;
  err_s0 = err_s0 * 100. ;
  err1 = err1 'ET' ( 'PROG' err_s0 ) ;
  err_e0 = 'ABS' ( epsp0 -  def_0 ) ;
  'SI' (def_0 '>' ( 1.E-1 * eps_el0 ) ) ;
     err_e0 = err_e0 / def_0 ;
  'SINON' ;
     err_e0 = 0. ;
  'FINSI' ;
  err_e0 = err_e0 * 100. ;
  'SI' ( err_e0 '>' 5.E-2 ) ;
       'ERRE' 5 ;
  'FINSI' ;
  'SI' ( err_s0 '>' 5.E-2 ) ;
       'ERRE' 5 ;
  'FINSI' ;
  err2 = err2 'ET' ( 'PROG' err_e0 ) ;
*
'FIN' bloc1 ;
*
*
'FIN' ;
```

Test preston1 **Comments**

- The Preston-Tonks-Wallace elastoplastic model

```
MODO = MODE VOL1 MECANIQUE ELASTIQUE ISOTROPE
       PLASTIQUE PRESTON ;
MATO = MATE MODO YOUN 2.1E11 NU 0.3
       RHO 7.8E3 TAU 0.025
       P 0. S0 0.0085 SINF 0.00055
       K 0. G 0.00001
       Y0 0.0001 YINF 0.0001 Y1 0.094 Y2 0.575
       BETA 0.25 GP 3.00 GT 0. MU 1.97
       TM0 1220. ;
```

Preston, Tonks and Wallace propose a mathematical formulation of the Von Mises's yielding stress based on the theory of the dislocations and the dimensional analysis. The constitutive equations of the model are:

-shear modulus G:
G = G0+GP'.P/(ETA**(1/3))+GT'.(T-300)
G0 = YOUNG0 / (2*(1+NU0))
where: YOUNG0 is the initial Young modulus
NU0 is the Poisson's ratio (stay constant)
G0 is the initial shear modulus
ETA is the compression rate: ETA=RHO/RHO0
TM = TM0 * EXP (2.MU0.(1.-1./ETA))/(ETA ** (2./3.))
and: if T>TM:
G = Y = 0 (Y is the yield limit)
then, the volumetric strain is fully elastic and the deviatoric strain is fully plastic

-Dimensionless terms:
Y' = Y/G
T' = T/TM
EPT' = EPT/X
with:
Y yielding stress
G shear modulus
T temperature
TM melting temperature
X = 1/6.(4/PI)**(.5).OMEGA
OMEGA: Debye's pulsation OMEGA = (G/RHO)**(.5)
RHO:material's density
EPT: equivalent total strain rate EPT=sqrt(2/3.ET:ET)
ET: total strain rate

-Dimensionless term for the saturation stress YS:
S1 = S0-(S0-SINF).erf(K'.T'.ln(g/EPT')
S2 = S0.(EPT'/g)**BETA
YS = max(S1,S2)

-Dimensionless term for the yielding limit YL:
L1 = Y0-(Y0-YINF).erf(K'.T'.ln(g/EPT')
L2 = Y1.(EPT'/g)**Y2
YL = max(L1,min(L2,S2))

-Dimensionless term for the yielding stress for Body Centered Cubic materials (B.C.C.): P=0
Y' = YS-(YS-YL).exp(-TAU.EP/(YS-YL))

-Dimensionless term for the yielding stress for the other materials: P differs from 0
Coeff1 = (S0-YL).(exp(P.(YS-YL)/(S0-YL))-1)
Coeff2 = 1-exp(-P.(YS-YL)/(S0-YL))
Y' = YS+(S0-YL)/P.ln(1-Coeff2.exp(-P.TAU.EP/Coeff1))

with:

EP: equivalent plastic strain
EP = sqrt(2/3.EPS:EPS)
EPS: palstic strain

The shear modulus G is expressed that way:
STRESS = HOOK.STRAIN i.e. D_STRESS = HOOK.D_STRAIN + D_HOOK.STRAIN

'RHO'	: initial density
'TAU'	: dimensionless parameter TAU used for the calculation of the hardening law
'P'	: dimensionless parameter P -if P=0, the microstucture of the material is BCC -else, the microstructure is NOT BCC
'S0'	: dimensionless parameter S0 it gives the saturation stress at T = 0K
'SINF'	: dimensionless parameter SINF it gives the saturation stress at T(K) infinite
'K'	: dimensionless term K'.T used for the calculation of the yielding limit. It is entered as an EVOLUTION object (the parameter is the temperature T)
'G'	: dimensionless parameter g used for the calculation of the saturation stress and the yielding limit
'Y0'	: dimensionless parameter Y0 it gives the yielding limit at T = $0K$
'YINF'	: dimensionless parameter YINF it gives the yielding limit at T(K) infinite
'Y1'	: dimensionless Y1used for the calculation of the yielding limit
'Y2'	: dimensionless Y2 used for the calculation of the yielding limit
'BETA'	: dimensionless parameter BETA used for the calculation of the yielding limit and the saturation stress
'GP'	: rate of the evolution of the shear modulus with the pressure (no unity): GP'
'GT'	: temperature-dependant term correcting the shear modulus (homogeneus to the shear modulus) One enters here the term: GT'.(T-300) with the use of an EVOLUTION object (the parameter of this EVOLUTION is the temperature).
'TMO'	: melting temperature for ETA = 1
'MU'	: coefficient MU0 used for the calculation of the melting temperature

Chapter 5

DAMAGE ELASTOPLASTIC MODELS

5.1 Test endocp1 Description sheet

Test name endocp1

Calculation type DAMAGE MODEL FOR ELASTOPLASTIC MATERIAL **Finite element type**

QUA8

Topic Tensile test on a damage elastoplastic material

The structure is an embedded test piece subjected to a tensile stress (imposed displacements). The structure obeys a law according to a damage elastoplastic model. **Goal** Test the model of damage material for MODE

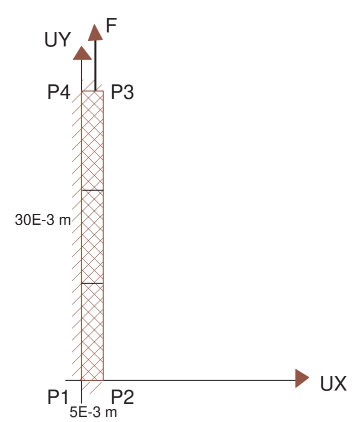
PLAN CONT and compare Castem's results with the analytical solution obtained by algorithm.
We also compute:
The cumulated plastic strain (EPSE component in VARIABLES_INTERNES).
The damage variable D in VARIABLES_INTERNES.
SIGMAyy (in CONTRAINTES).

Reference Analytical solution.
Tests carried out by Mr STRUB (rapport DMT/91 .558).

Version 97' customer version

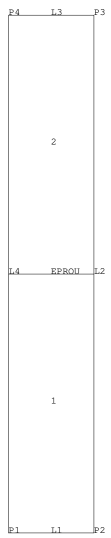
Model description

E	=	$74020\ 10^6$
NU	=	0.3
RHO	=	7800.
ALPH	=	0.
DC	=	0.23
EPSD	=	0.03
EPSR	=	0.25

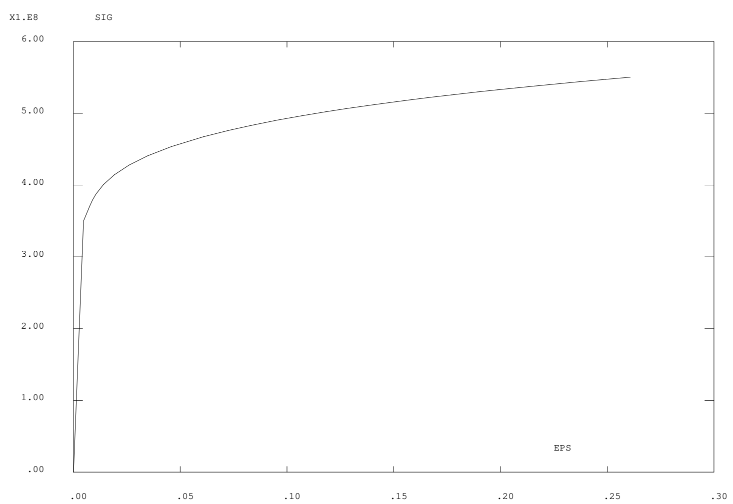


Test endocp1 **Results**

CASTEM FIGURES



DAMAGE ELASTOPLASTIC MODEL



DAMAGE ELASTOPLASTIC MODEL

```
*          Test Endocp1.dgibi: Jeux de données      *
*          -----------------------------------      *
*                                                   *
*
complet = faux;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
*************************************************
**
*******************************************************
*                                                     *
* TEST;  MATERIAU ELASTO-PLASTIQUE ENDOMMAGEABLE      *
*                                                     *
* EPROUVETTE EN TRACTION AVEC DEPLACEMENTS IMPOSES.   *
* MAILLAGE AXISYMETRIQUE.                             *
*                                                     *
* LES RESULTATS OBTENUS SONT COMPARES  A LA SOLUTION  *
* ANALYTIQUE OBTENUE PAR ALGORITHME.                  *
* ON TESTE - LA DEFORMATION PLASTIQUE CUMULEE         *
*            (composante EPSE dans VARIABLES_INTERNES)*
*          - D, LA VARIABLE D'ENDOMMAGEMENT           *
*            dans VARIABLES_INTERNES).                *
*          - SIGMAyy ( dans CONTRAINTES ).            *
*                                                     *
* Test effectue par Mr STRUB (cf rapport DMT/91 .558).*
* Test modifie pour evolution 2/06/92.                *
*******************************************************

GRAPH = 'N';
OPTI ECHO 0 DIME 2 ELEM QUA8 MODE PLAN CONT ;
*
*-----------------------------------------------------
   P1 = 0 0;          P2 = 5E-3 0;
   P3 = 5E-3 30E-3;   P4 = 0 30E-3;
*
   L1 = P1 P2 DROIT 1  ;
   L2 = P2 P3 DROIT 2 ;
   L3 = P3 P4 DROIT 1  ;
   L4 = P4 P1 DROIT 2 ;
*
   EPROU = L1 L2 L3 L4 DALLER PLAN ;

SI ( NEG GRAPH 'N');
   trac eprou;
finsi;
   CL1 = BLOQ L1 UY ;
   CL2 = BLOQ L3 UY ;
   CL3 = BLOQ L4 UX ;
   CL = CL1 ET CL2 ET CL3 ;
   DEP = DEPI CL2 1E-3 ;
*
   MO = MODL EPROU 'MECANIQUE' 'ELASTIQUE' 'PLASTIQUE'
        'ENDOMMAGEABLE';
*
  PROGEPS = PROG 0.    .47285E-2 .76066E-2 .89226E-2
   1.0533E-2 1.4011E-2 1.9096E-2 2.6081E-2 3.4654E-2
   4.5828E-2 6.0813E-2 7.2835E-2 8.3328E-2 9.5526E-2
   10.691E-2 11.808E-2 12.784E-2 13.891E-2 14.936E-2
   15.700E-2 16.746E-2 17.720E-2 18.946E-2 19.860E-2
   20.703E-2 21.607E-2 22.802E-2 23.685E-2 24.919E-2
   26.083E-2 ;
  PROGSIG = PROG 0.    350.E06   370.59E06 379.18E06
   387.37E06 400.52E06 414.22E06 427.94E06 440.69E06
   453.63E06 467.26E06 476.30E06 483.24E06 490.46E06
   496.56E06 502.06E06 506.53E06 511.28E06 515.51E06
   518.45E06 522.29E06 525.71E06 529.81E06 532.73E06
   535.33E06 538.03E06 541.47E06 543.92E06 547.23E06
   550.24E06 ;
   CTRAC = EVOL MANU EPS PROGEPS SIG PROGSIG ;
*
SI ( NEG GRAPH 'N');
    DESS CTRAC;
FINSI;
  MAT = MATR MO YOUN 74020.E06 NU 0.3 RHO 7800. ALPH 0.
        TRAC CTRAC EPSD 0.03 DC 0.23 EPSR 0.25 ;
*
   LI1 = PROG 0. 30. ; LI2 = PROG 0. 30. ;
   EV = EVOL MANU T LI1 DEP(T) LI2 ;
   CHA = CHAR 'DIMP' DEP EV ;
*
* LISTE DES PAS DE CALCUL :
*                        0.13 MM INCREMENT ELASTIQUE
*                        0.16 MM INCREMENT PLASTIQUE
si complet;
 LIS = PROG 0.13 0.16 0.33 0.5 0.75 1. 1.25 1.5 1.75 2.
           2.25 2.5 2.75 3. 3.25 3.5 3.75 4. 4.25 4.5
           4.75 5. 5.25 5.5 5.75 6. 6.25 6.5 6.75 7.
           7.25 7.5 7.75 8. ;
sinon;
 LIS = prog  0.13 0.16  0.33 0.5 ;
finsi;
    TAB = TABLE ;
    TAB.'BLOCAGES_MECANIQUES' = CL;
    TAB.'MODELE' = MO;
    TAB.'CARACTERISTIQUES' = MAT;
    TAB.'TEMPS_CALCULES' = LIS;
    TAB.'CHARGEMENT' = CHA;
    TAB.'PRECISION' = 1E-08;
*
    PASAPAS TAB;

*******************************************************
************     TEST DE FONCTIONNEMENT  **************
*******************************************************

dpc0 = extr TAB.VARIABLES_INTERNES.2 EPSE 1 1 1;
dpc1 = extr TAB.VARIABLES_INTERNES.4 EPSE 1 1 1;
dpc0t = 5.46e-4; dpc1t =1.12694E-02;
si complet;
dpc2 = extr TAB.VARIABLES_INTERNES.10 EPSE 1 1 1;
dpc4 = extr TAB.VARIABLES_INTERNES.18 EPSE 1 1 1;
dpc6 = extr TAB.VARIABLES_INTERNES.26 EPSE 1 1 1;
dpc75 =extr TAB.VARIABLES_INTERNES.32 EPSE 1 1 1;
dpc2t = 6.03e-2; dpc4t = 12.65e-2; dpc6t = 19.29e-2;
dpc75t = 24.27e-2;
finsi;


D0 =  extr TAB.VARIABLES_INTERNES.2 'VHWD' 1 1 1;
D1 =  extr TAB.VARIABLES_INTERNES.4 'VHWD' 1 1 1;
D0T = 0.; D1T= 0.0;
si complet;
D2 =  extr TAB.VARIABLES_INTERNES.10 'VHWD' 1 1 1;
D4 =  extr TAB.VARIABLES_INTERNES.18 'VHWD' 1 1 1;
D6 =  extr TAB.VARIABLES_INTERNES.26 'VHWD' 1 1 1;
D75 =  extr TAB.VARIABLES_INTERNES.32 'VHWD' 1 1 1;
D2T= 0.0257;  D4T= 0.0901; D6T= 0.1622; D75T= 0.2199;
finsi;

SZZ0 = EXTR TAB.CONTRAINTES.2 SMYY 1 1 1;
SZZ1 = EXTR TAB.CONTRAINTES.4 SMYY 1 1 1;
SZZ0T = 354.3E6; SZZ1T =  4.07984E+08;
si complet;
SZZ2 = EXTR TAB.CONTRAINTES.10 SMYY 1 1 1;
SZZ4 = EXTR TAB.CONTRAINTES.18 SMYY 1 1 1;
```

```
SZZ6 = EXTR TAB.CONTRAINTES.26 SMYY 1 1 1;
SZZ75 = EXTR TAB.CONTRAINTES.32 SMYY 1 1 1;
 SZZ2T = 459.3E6; SZZ4T = 461.4E6; SZZ6T = 443.3E6;
SZZ75T = 422.4E6;
finsi;

EDPC0 = ABS((DPC0-DPC0T)/DPC0T);
EDPC1 = ABS((DPC1-DPC1T)/DPC1T);
si complet;
EDPC2 = ABS((DPC2-DPC2T)/DPC2T);
EDPC4 = ABS((DPC4-DPC4T)/DPC4T);
EDPC6 = ABS((DPC6-DPC6T)/DPC6T);
EDPC75 = ABS((DPC75-DPC75T)/DPC75T);
finsi;
ED0 = ABS((D0-D0T)/(D0T+0.001));
ED1 = ABS((D1-D1T)/(D1T+0.001));
si complet;
ED2 = ABS((D2-D2T)/D2T);
ED4 = ABS((D4-D4T)/D4T);
ED6 = ABS((D6-D6T)/D6T);
ED75 = ABS((D75-D75T)/D75T);
finsi;
ESZZ0 = ABS((SZZ0-SZZ0T)/SZZ0T);
ESZZ1 = ABS((SZZ1-SZZ1T)/SZZ1T);
si complet;
ESZZ2 = ABS((SZZ2-SZZ2T)/SZZ2T);
ESZZ4 = ABS((SZZ4-SZZ4T)/SZZ4T);
ESZZ6 = ABS((SZZ6-SZZ6T)/SZZ6T);
ESZZ75 = ABS((SZZ75-SZZ75T)/SZZ75T);
finsi;
si complet;

LERR = PROG EDPC0 EDPC1 EDPC2 EDPC4 EDPC6 EDPC75 ED0
       ED1 ED2 ED4 ED6 ED75
       ESZZ0 ESZZ1 ESZZ2 ESZZ4 ESZZ6 ESZZ75;
sinon;
lerr =  PROG EDPC0 EDPC1 ED0 ED1 ESZZ0 ESZZ1;
finsi;
list LERR;
ermax = maxi LERR;
LIST ERMAX;

temps;
SI ( ERMAX <EG 0.05 );
   ERRE 0;
SINON;
   ERRE 5;
FINSI;

FIN;
```

Test endocp1 **Comments**

- The damage model for elastoplastic material (Lemaitre-Chaboche)

```
MO = MODE EPROU MECANIQUE ELASTIQUE PLASTIQUE
          ENDOMMAGEABLE;
MAT = MATE MO YOUN 74020.E06 NU 0.3 RHO 7800.
           ALPH 0. TRAC CTRAC EPSD 0.03
           DC 0.23 EPSR 0.25 ;
```

The Lemaitre-Chaboche model assume that stress hardening and damage are isotropic. The Von Mises' criterion is coupled with damage. The following parameters will be specified BESIDES those relative to the elastic behavior.

'TRAC'	: Key word followed by:
NOMTRAC	: Tension curve consisting of an EVOLUTIO type object with the strains in abscissa and the stresses in ordinate. It must contain the origin. It may be drawn by means of the DESSINE instruction.
'EPSD'	: Damage threshold: it is the plastic deformation above which the material gets damaged.
'DC '	: Critical value of the D variable describing damage. DC features the material rupture.
'EPSR'	: Plastic deformation corresponding to material rupture.

5.2 Test endoaxi1 Description sheet

Test name endoaxi1

Calculation type DAMAGE MODEL FOR ELASTOPLASTIC MATERIAL **Finite element type**

QUA8

Topic Tensile test on a damage elastoplastic material

The structure is an embedded test piece subjected to a tensile stress (imposed displacements). The structure obeys a law according to a damage elastoplastic model. **Goal** Test the model of damage material for MODE

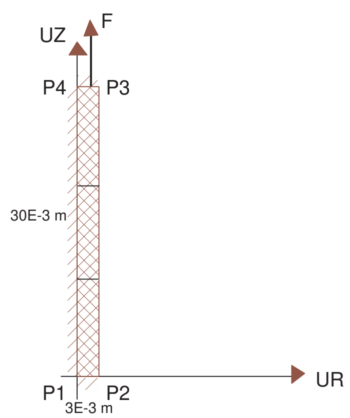
AXIS and compare Castem's results with the analytical solution obtained by algorithm.
We also compute:
The cumulated plastic strain (EPSE component in VARIABLES_INTERNES).
The damage variable D in VARIABLES_INTERNES.
SIGMAzz (in CONTRAINTES).

Reference Analytical solution.
Tests carried out by Mr STRUB (rapport DMT/91 .558).

Version 97' customer version

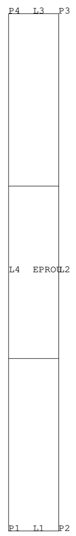
Model description

$$
\begin{aligned}
E &= 74020\ 10^6 \\
NU &= 0.3 \\
RHO &= 7800. \\
ALPH &= 0. \\
DC &= 0.23 \\
EPSD &= 0.03 \\
EPSR &= 0.25
\end{aligned}
$$

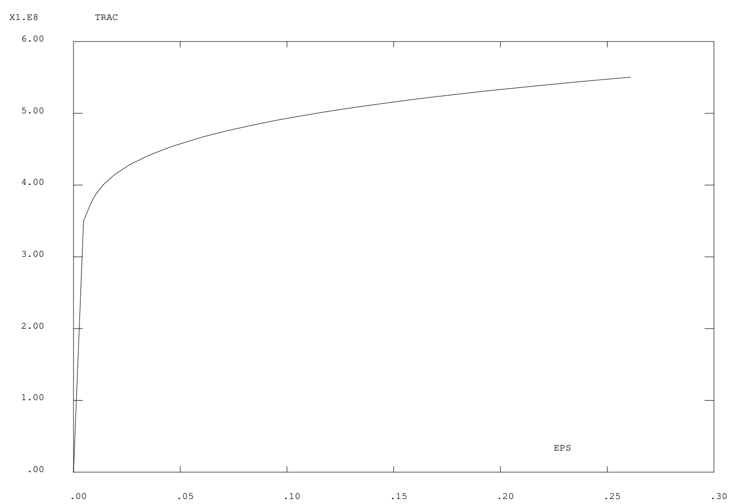


Test endoaxi1 **Results**

CASTEM FIGURES



DAMAGE ELASTOPLASTIC MODEL



DAMAGE ELASTOPLASTIC MODEL

```
*          Test Endoaxi1.dgibi: Jeux de données     *
*          -----------------------------------      *
*                                                   *
*
complet = faux;
*
**************************************************
*  pour calcul complet mettre complet à : vrai;
**************************************************
********************************************************
*                                                    *
* TEST;  MATERIAU ELASTO-PLASTIQUE ENDOMMAGEABLE      *
*                                                    *
* EPROUVETTE EN TRACTION AVEC DEPLACEMENTS IMPOSES.  *
* MAILLAGE AXISYMETRIQUE.                            *
*                                                    *
* LES RESULTATS OBTENUS SONT COMPARES  A LA SOLUTION *
* ANALYTIQUE OBTENUE PAR ALGORITHME.                 *
* ON TESTE - LA DEFORMATION PLASTIQUE CUMULEE        *
*           (composante EPSE dans VARIABLES_INTERNES)*
*         - D, LA VARIABLE D'ENDOMMAGEMENT           *
*           (DANS VARIABLES_INTERNES).               *
*         - SIGMAzz ( dans CONTRAINTES ).            *
*                                                    *
* Test effectue par Mr STRUB (cf rapport DMT/91 .558).*
* Test modifie pour evolution 2/06/92.               *
********************************************************

GRAPH = 'N';

OPTI ECHO 1 DIME 2 ELEM QUA8 MODE AXIS;
opti trac bens;
*------------------------------------------------------
   P1 = 0 0;          P2 = 3E-3 0;
   P3 = 3E-3 30E-3;   P4 = 0 30E-3;
*
   L1 = P1 P2 DROIT 1  ;
   L2 = P2 P3 DROIT 3 ;
   L3 = P3 P4 DROIT 1  ;
   L4 = P4 P1 DROIT 3 ;
*
   EPROU = L1 L2 L3 L4 DALLER PLAN ;

SI ( NEG GRAPH 'N');
   trac eprou;
FINSI;

   CL1 = BLOQ L1 UZ ;
   CL2 = BLOQ L3 UZ ;
   CL3 = BLOQ L4 UR ;
   CL = CL1 ET CL2 ET CL3 ;
   DEP = DEPI CL2 1E-3 ;
*
   MO = MODL EPROU 'MECANIQUE' 'ELASTIQUE' 'PLASTIQUE'
'ENDOMMAGEABLE';
*
  PROGEPS = PROG 0. .47285E-2 .76066E-2 .89226E-2
   1.0533E-2 1.4011E-2 1.9096E-2 2.6081E-2 3.4654E-2
   4.5828E-2 6.0813E-2 7.2835E-2 8.3328E-2 9.5526E-2
   10.691E-2 11.808E-2 12.784E-2 13.891E-2 14.936E-2
   15.700E-2 16.746E-2 17.720E-2 18.946E-2 19.860E-2
   20.703E-2 21.607E-2 22.802E-2 23.685E-2 24.919E-2
   26.083E-2 ;
  PROGSIG = PROG 0. 350.E06 370.59E06 379.18E06
   387.37E06 400.52E06 414.22E06 427.94E06 440.69E06
   453.63E06 467.26E06 476.30E06 483.24E06 490.46E06
   496.56E06 502.06E06 506.53E06 511.28E06 515.51E06
   518.45E06 522.29E06 525.71E06 529.81E06 532.73E06
   535.33E06 538.03E06 541.47E06 543.92E06 547.23E06
   550.24E06 ;
   CTRAC = EVOL MANU EPS PROGEPS TRAC PROGSIG ;

SI ( NEG GRAPH 'N');
   DESS CTRAC;
FINSI;

  MAT = MATR MO YOUN 74020.E06 NU 0.3 RHO 7800. ALPH 0.
               TRAC CTRAC EPSD 0.03 DC 0.23 EPSR 0.25 ;
*
* LISTE DES PAS DE CALCUL :
*                        0.13 MM INCREMENT ELASTIQUE
*                        0.16 MM INCREMENT PLASTIQUE
si complet;
    LIS = PROG 0.13 0.16
               0.5 1. 1.25 1.5 2. 2.5 3. 3.5 4. 4.5
               5. 5.5 6. 6.5 7. 7.5 8.;
sinon;
     LIS = prog  0.13 0.16  0.5 ;
finsi;
*
    LI1 = PROG 0. 30. ; LI2 = PROG 0. 30. ;
    EV = EVOL MANU T LI1 DEP(T) LI2 ;
    CHA = CHAR 'DIMP' DEP EV ;
    TAB = TABLE ;

    TAB.'BLOCAGES_MECANIQUES' = CL;
    TAB.'CARACTERISTIQUES' = MAT;
    TAB.'CHARGEMENT' = CHA;
    TAB.'MODELE' = MO;
    TAB.'TEMPS_CALCULES' = LIS;
    TAB.'PRECISION' = 1E-08;
*
 PASAPAS  TAB ;

*******************************************************
***************    TEST DE FONCTIONNEMENT  ***********
*******************************************************

dpc0 = extr TAB.VARIABLES_INTERNES.2 EPSE 1 1 1;
dpc1 = extr TAB.VARIABLES_INTERNES.3 EPSE 1 1 1;
dpc0t = 5.46e-4;dpc1t =1.12694E-02;
si complet;
dpc2 = extr TAB.VARIABLES_INTERNES.7 EPSE 1 1 1;
dpc4 = extr TAB.VARIABLES_INTERNES.11 EPSE 1 1 1;
dpc6 = extr TAB.VARIABLES_INTERNES.15 EPSE 1 1 1;
dpc75 =extr TAB.VARIABLES_INTERNES.18 EPSE 1 1 1;

 dpc2t = 6.03e-2; dpc4t = 12.65e-2; dpc6t = 19.29e-2;
dpc75t = 24.27e-2;
finsi;
D0 =  extr TAB.VARIABLES_INTERNES.2 'VHWD' 1 1 1;
D1 =  extr TAB.VARIABLES_INTERNES.3 'VHWD' 1 1 1;
D0T = 0.; D1T= 0.0;
si complet;
D2 =  extr TAB.VARIABLES_INTERNES.7 'VHWD' 1 1 1;
D4 =  extr TAB.VARIABLES_INTERNES.11 'VHWD' 1 1 1;
D6 =  extr TAB.VARIABLES_INTERNES.15 'VHWD' 1 1 1;
D75 =  extr TAB.VARIABLES_INTERNES.18 'VHWD' 1 1 1;

 D2T= 0.0257; D4T= 0.0901; D6T= 0.1622; D75T= 0.2199;
  finsi;

SZZ0 = EXTR TAB.CONTRAINTES.2 SMZZ 1 1 1;
SZZ1 = EXTR TAB.CONTRAINTES.3 SMZZ 1 1 1;
SZZ0T = 354.3E6; SZZ1T =  4.07984E+08;
si complet;
```

```
SZZ2 = EXTR TAB.CONTRAINTES.7 SMZZ 1 1 1;
SZZ4 = EXTR TAB.CONTRAINTES.11 SMZZ 1 1 1;
SZZ6 = EXTR TAB.CONTRAINTES.15 SMZZ 1 1 1;
SZZ75 = EXTR TAB.CONTRAINTES.18 SMZZ 1 1 1;

SZZ2T = 459.3E6; SZZ4T = 461.4E6; SZZ6T = 443.3E6;
SZZ75T = 422.4E6;
finsi;

EDPC0 = ABS((DPC0-DPC0T)/DPC0T);
EDPC1 = ABS((DPC1-DPC1T)/DPC1T);
si complet;
EDPC2 = ABS((DPC2-DPC2T)/DPC2T);
EDPC4 = ABS((DPC4-DPC4T)/DPC4T);
EDPC6 = ABS((DPC6-DPC6T)/DPC6T);
EDPC75 = ABS((DPC75-DPC75T)/DPC75T);
finsi;
ED0 = ABS((D0-D0T)/(D0T+0.001));
ED1 = ABS((D1-D1T)/(D1T+0.001));
si complet;
ED2 = ABS((D2-D2T)/D2T);
ED4 = ABS((D4-D4T)/D4T);
ED6 = ABS((D6-D6T)/D6T);
ED75 = ABS((D75-D75T)/D75T);
finsi;

ESZZ0 = ABS((SZZ0-SZZ0T)/SZZ0T);
ESZZ1 = ABS((SZZ1-SZZ1T)/SZZ1T);
si complet;
ESZZ2 = ABS((SZZ2-SZZ2T)/SZZ2T);
ESZZ4 = ABS((SZZ4-SZZ4T)/SZZ4T);
ESZZ6 = ABS((SZZ6-SZZ6T)/SZZ6T);
ESZZ75 = ABS((SZZ75-SZZ75T)/SZZ75T);

finsi;
si complet;
LERR = PROG EDPC0 EDPC1 EDPC2 EDPC4 EDPC6 EDPC75 ED0
 ED1 ED2 ED4 ED6 ED75
       ESZZ0 ESZZ1 ESZZ2 ESZZ4 ESZZ6 ESZZ75;
sinon;
lerr =  PROG EDPC0 EDPC1 ED0 ED1 ESZZ0 ESZZ1;
finsi;
list LERR;

ermax = maxi LERR;
LIST ermax;
temps;
SI ( ERMAX <EG 0.05 );
   ERRE 0;
SINON;
   ERRE 5;
FINSI;

FIN;
 FIN;
```

5.3 Test endoaxi2 Description sheet

Test name endoaxi2

Calculation type DAMAGE MODEL FOR ELASTOPLASTIC MATERIAL

Finite element type QUA8

Topic Tensile test and thermical load on a damage elastoplastic material

The structure is an embedded test piece subjected to a tensile stress (imposed displacements) and a thermical load. The structure obeys a law according to a damage elastoplastic model. Furthermore the behaviour of the material depends on the temperarure through its tensile curve and thermical expansion coefficient. **Goal**

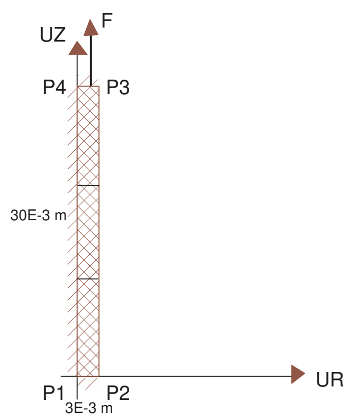
Test the damage material model for MODE AXIS and compare Castem's results with the analytical solution obtained by algorithm.
We also compute:
The cumulated plastic strain (EPSE component in VARIABLES_INTERNES).
The damage variable D in VARIABLES_INTERNES.
SIGMAzz (in CONTRAINTES).

Reference Analytical solution.

Version 97' customer version

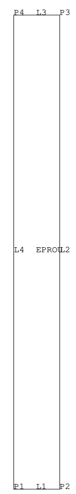
Model description

E	=	$74020\ 10^6$
NU	=	0.3
RHO	=	7800.
ALPH	=	variable
DC	=	0.23
EPSD	=	0.
EPSR	=	0.25

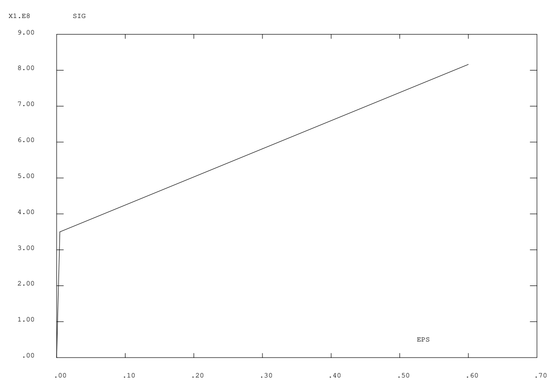


Test endoaxi2 Results

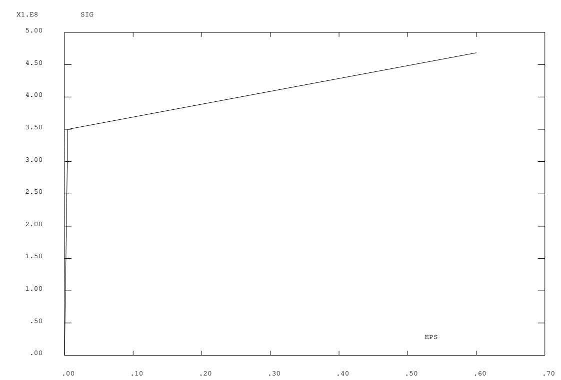
CASTEM FIGURES



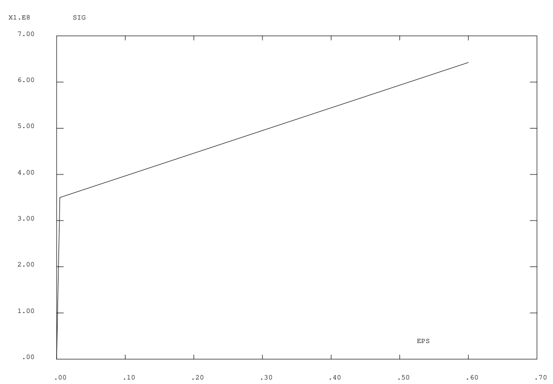
DAMAGE ELASTOPLASTIC MODEL



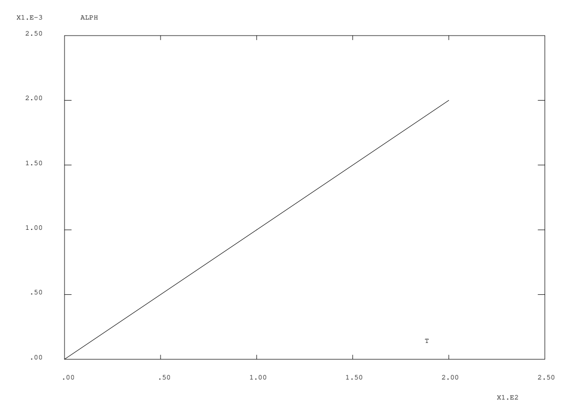
DAMAGE ELASTOPLASTIC MODEL



DAMAGE ELASTOPLASTIC MODEL



DAMAGE ELASTOPLASTIC MODEL



DAMAGE ELASTOPLASTIC MODEL

```
*        Test Endoaxi2.dgibi: Jeux de données       *
*        -----------------------------------        *
*                                                   *
*****************************************************
*                                                   *
* TEST;  MATERIAU ELASTO-PLASTIQUE ENDOMMAGEABLE    *
* MATERIAU DEPENDANT DE LA TEMPERATURE :            *
*                          COURBE DE TRACTION       *
*                          COEFFICIENT DE DILATATION *
*                          THERMIQUE                *
*                                                   *
* EPROUVETTE EN TRACTION AVEC DEPLACEMENTS IMPOSES  *
*                           + CHARGEMENT THERMIQUE  *
* MAILLAGE AXISYMETRIQUE.                           *
*                                                   *
* LES RESULTATS OBTENUS SONT COMPARES  A LA SOLUTION *
* ANALYTIQUE OBTENUE PAR ALGORITHME.                *
* ON TESTE - LA DEFORMATION PLASTIQUE CUMULEE       *
*           (composante EPSE dans VARIABLES_INTERNES)*
*         - D, LA VARIABLE D'ENDOMMAGEMENT          *
*           DANS VARIABLES_INTERNES .               *
*         - SIGMAzz ( DANS CONTRAINTES ).           *
*****************************************************


OPTI ECHO 0 DIME 2 ELEM QUA8 MODE AXIS;
*-----------------------------------------------------
   P1 = 0 0;          P2 = 3E-3 0;
   P3 = 3E-3 30E-3;   P4 = 0 30E-3;
*
   L1 = P1 P2 DROIT 1  ;
   L2 = P2 P3 DROIT 1  ;
   L3 = P3 P4 DROIT 1  ;
   L4 = P4 P1 DROIT 1  ;
*
   EPROU = L1 L2 L3 L4 DALLER PLAN ;

   CL1 = BLOQ L1 UZ ;
   CL2 = BLOQ L3 UZ ;
   CL3 = BLOQ L4 UR ;
   CL = CL1 ET CL2 ET CL3 ;
   DEP = DEPI CL2 1E-3 ;
*
   MO = MODL EPROU 'MECANIQUE' 'ELASTIQUE' 'PLASTIQUE'
'ENDOMMAGEABLE';
*
* module tangent valant 783MPa
*-----------------------------
  PROGEPS1 = PROG 0. (350./74020.) 0.3 0.6 ;
  PROGSIG1 = PROG 0. 350.E06 581.1976E06 816.0976E06 ;
   CTRAC1 = EVOL MANU EPS PROGEPS1 SIG PROGSIG1 ;
*
* module tangent valant 491MPa
*-----------------------------
  PROGEPS2 = PROG 0. (350./74020.) 0.3 0.6 ;
  PROGSIG2 = PROG 0. 350.E06 494.9783E06 642.2783E06 ;
   CTRAC2 = EVOL MANU EPS PROGEPS2 SIG PROGSIG2 ;
*
* module tangent valant 199MPa
*-----------------------------
  PROGEPS3 = PROG 0. (350./74020.) 0.3 0.6 ;
  PROGSIG3 = PROG 0. 350.E06 408.7590E06 468.4590E06 ;
   CTRAC3 = EVOL MANU EPS PROGEPS3 SIG PROGSIG3 ;
*
* module tangent variable
*------------------------
CTRAC = NUAGE 'T'*'FLOTTANT' 'TRAC'*'EVOLUTION'
                  0. CTRAC1 100. CTRAC2 200. CTRAC3  ;
*
* coefficient de dilatation thermique vaiable
*--------------------------------------------
*
  PROGTEMP = PROG 0. 100. 200. ;
  PROGALFA = PROG 0. 1E-3 2E-3 ;
  EVALPH = EVOL MANU 'T' PROGTEMP 'ALPH' PROGALFA ;
*
*   MAT = MATR MO YOUN 74020.E06 NU 0.3 RHO 7800.
*   ALPH 1.E-3 TRAC CTRAC1 EPSD 0. DC 0.23 EPSR 0.25 ;
*
   MATvar = MATR MO YOUN 74020.E06 NU 0.3 RHO 7800.
   ALPH EVALPH TRAC CTRAC EPSD 0. DC 0.23 EPSR 0.25 ;
*
************** DEFINITION DU CHARGEMENT ***************
*
   LI1 = PROG 0. 30. ; LI2 = PROG 0. 30. ;
   EV = EVOL MANU T LI1 DEP(T) LI2 ;
   CHA1 = CHAR 'DIMP' DEP EV ;
   TEMP0 = MANU CHPO EPROU 1 'T' 0.      ;
   TEMP200 = MANU CHPO EPROU 1 'T' 200. ;
   t0 = 0. ; t20 = 20.      ;
   TEMP = TABLE; TEMPERA = TABLE;                       *
   TEMP.0 = t0 ; TEMP.1 = t20 ;
   TEMPERA.0 = TEMP0; TEMPERA.1 = TEMP200;
   CHA2 = CHAR 'T' TEMP TEMPERA;
   CHA = CHA1 ET CHA2;
*
* LISTE DES PAS DE CALCUL :
*                        0.14 MM INCREMENT ELASTIQUE
*                        0.15 MM INCREMENT PLASTIQUE
  LIS = PROG 0.14 0.15 0.2 0.4 0.7 1 pas .3
       3.1 3.3 pas .2 4.1 ;
*
TAB = TABLE ;
TAB.'PRECISION' = 1E-3;
TAB.'BLOCAGES_MECANIQUES' = CL;
TAB.'CARACTERISTIQUES' = MATvar;
TAB.'CHARGEMENT' = CHA;
TAB.'TEMPS_CALCULES' = LIS;
TAB.'MODELE' = MO;
PASAPAS TAB ;
*
*
*-----------------------------------------------------
*  CONTROLE DES RESULTATS AVEC DE LA SOLUTION
*      DE REFERENCE OBTENUE PAR ALGORITHME
*-----------------------------------------------------
*
 REF_D = PROG 2.197E-2 3.417E-2 6.337E-2 8.131E-2 ;
 REF_P = PROG 3.646E-2 5.443E-2 9.296E-2 1.143E-1 ;
 REF_S = (PROG   368.9   376.0   386.4   389.2)*1E6 ;
*
D1 = EXTR (TAB.VARIABLES_INTERNES.7) 'VHWD' 1 1 1;
D2 = EXTR (TAB.VARIABLES_INTERNES.9) 'VHWD' 1 1 1;
D3 = EXTR (TAB.VARIABLES_INTERNES.14) 'VHWD' 1 1 1;
D4 = EXTR (TAB.VARIABLES_INTERNES.18) 'VHWD' 1 1 1;
*
P1 = EXTR (TAB.VARIABLES_INTERNES.7) 'EPSE' 1 1 1;
P2 = EXTR (TAB.VARIABLES_INTERNES.9) 'EPSE' 1 1 1;
P3 = EXTR (TAB.VARIABLES_INTERNES.14) 'EPSE' 1 1 1;
P4 = EXTR (TAB.VARIABLES_INTERNES.18) 'EPSE' 1 1 1;
*
S1 = EXTR (TAB.CONTRAINTES.7) 'SMZZ' 1 1 1;
S2 = EXTR (TAB.CONTRAINTES.9) 'SMZZ' 1 1 1;
S3 = EXTR (TAB.CONTRAINTES.14) 'SMZZ' 1 1 1;
S4 = EXTR (TAB.CONTRAINTES.18) 'SMZZ' 1 1 1;
```

```
*
EF_D = PROG D1 D2 D3 D4 ;
EF_P = PROG P1 P2 P3 P4 ;
EF_S = PROG S1 S2 S3 S4 ;
*
PROG_ERD = MAXI (ABS ((EF_D  - REF_D) / REF_D)) ;
PROG_ERP = MAXI (ABS ((EF_P  - REF_P) / REF_P)) ;
PROG_ERS = MAXI (ABS ((EF_S  - REF_S) / REF_S)) ;
*
ERR = MAXI (PROG PROG_ERD PROG_ERP PROG_ERS)    ;
*
temps;
SI ( ERR <EG 0.05 );
   ERRE 0;
SINON;
   ERRE 5;
FINSI;
*
FIN;
```

5.4 Test endoaxi3 Description sheet

Test name endoaxi3

Calculation type DAMAGE MODEL FOR ELASTOPLASTIC MATERIAL

Finite element type QUA8

Topic Tensile test and thermical load on a damage elastoplastic material

The structure is an embedded test piece subjected to a tensile stress (imposed displacements) and a thermical load. The structure obeys a law according to a damage elastoplastic model. Furthermore the behaviour of the material depends on temperarure through its tensile curve and thermical expansion coefficient.

Reference temperature	=	30^oC
Reference temperature for the thermic expansion coefficients	=	10^oC
Initial temperature	=	50^oC

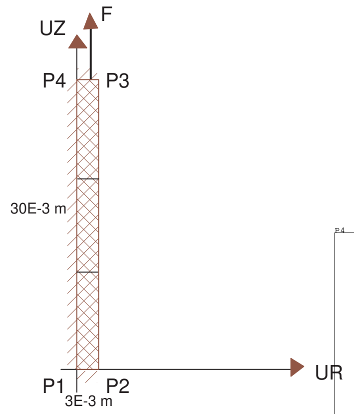
Goal Test the damage material model for MODE AXIS and compare Castem's results with the analytical solution obtained by algorithm.
We also compute:
The cumulated plastic strain (EPSE component in VARIABLES_INTERNES).
The damage variable D in VARIABLES_INTERNES.
SIGMAzz (in CONTRAINTES).

Reference Analytical solution.

Version 97' customer version

Model description

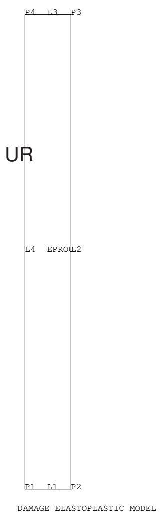
E	=	$74020\ 10^6$
NU	=	0.3
RHO	=	7800.
ALPH	=	variable
DC	=	0.23
EPSD	=	0.
EPSR	=	0.25

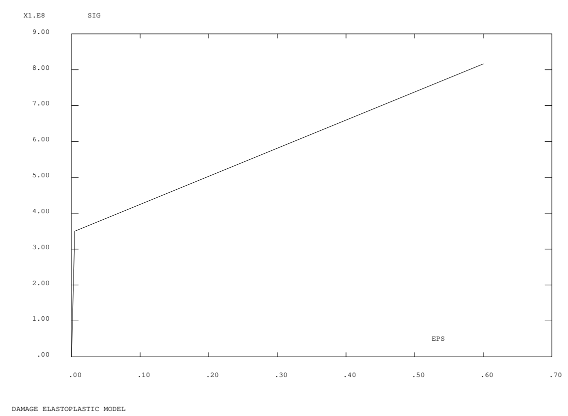


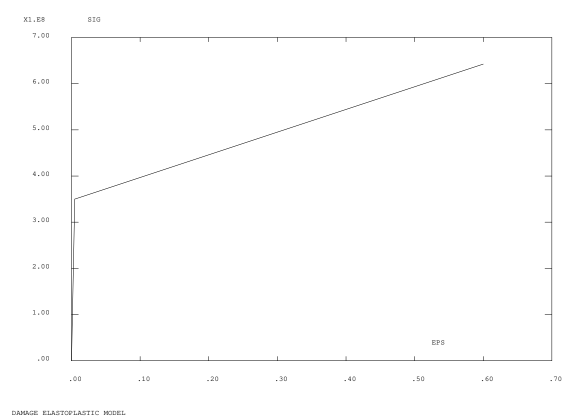
Test endoaxi3

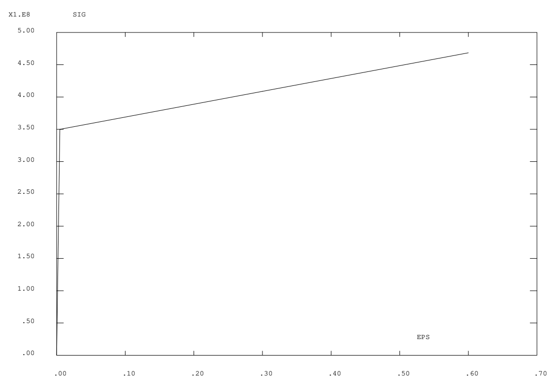
Results

CASTEM FIGURES

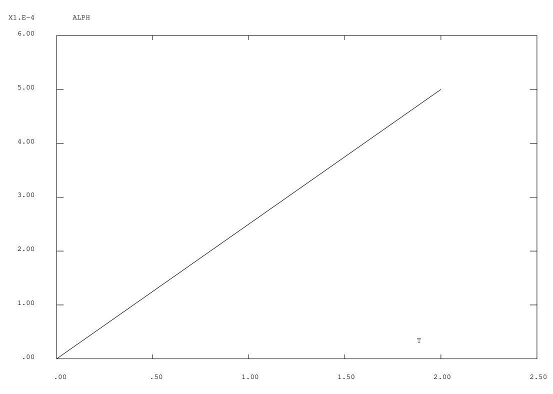








DAMAGE ELASTOPLASTIC MODEL



DAMAGE ELASTOPLASTIC MODEL

```
*          Test Endoaxi3.dgibi: Jeux de données      *
*          ------------------------------------      *
*                                                    *
complet = faux;
*
**************************************************
*  pour calcul complet mettre complet à : vrai;
****************************************************
******************************************************
*                                                    *
* TEST;  MATERIAU ELASTO-PLASTIQUE ENDOMMAGEABLE     *
* MATERIAU DEPENDANT DE LA TEMPERATURE :             *
*                         COURBE DE TRACTION         *
*                         COEFFICIENT DE DILATATION  *
*                         THERMIQUE                  *
* TEMPERATURE DE REFERENCE = 30C                    *
* TEMPERATURE DE REFERENCE POUR COEF. DE DILATATION  *
* THERMIQUES              = 10C                     *
* TEMPERATURE INITIALE    = 50C                     *
*                                                    *
* EPROUVETTE EN TRACTION AVEC DEPLACEMENTS IMPOSES   *
*                          + CHARGEMENT THERMIQUE    *
* MAILLAGE AXISYMETRIQUE.                            *
*                                                    *
* LES RESULTATS OBTENUS SONT COMPARES  A LA SOLUTION *
* ANALYTIQUE OBTENUE PAR ALGORITHME.                 *
* ON TESTE - LA DEFORMATION PLASTIQUE CUMULEE        *
*           (composante EPSE dans VARIABLES_INTERNES)*
*          - D, LA VARIABLE D'ENDOMMAGEMENT          *
*            DANS VARIABLES_INTERNES .               *
*          - SIGMAzz ( DANS CONTRAINTES ).           *
******************************************************


OPTI ECHO 0 DIME 2 ELEM QUA8 MODE AXIS;
*-----------------------------------------------------
   P1 = 0 0;          P2 = 3E-3 0;
   P3 = 3E-3 30E-3;   P4 = 0 30E-3;
*
   L1 = P1 P2 DROIT 1  ;
   L2 = P2 P3 DROIT 1  ;
   L3 = P3 P4 DROIT 1  ;
   L4 = P4 P1 DROIT 1  ;
*
   EPROU = L1 L2 L3 L4 DALLER PLAN ;

   CL1 = BLOQ L1 UZ ;
   CL2 = BLOQ L3 UZ ;
   CL3 = BLOQ L4 UR ;
   CL = CL1 ET CL2 ET CL3 ;
   DEP = DEPI CL2 -1E-3 ;
*
   MO = MODL EPROU 'MECANIQUE' 'ELASTIQUE' 'PLASTIQUE'
'ENDOMMAGEABLE';
*
* module tangent valant 783MPa
*-----------------------------
  PROGEPS1 = PROG 0. (350./74020.) 0.3 0.6 ;
  PROGSIG1 = PROG 0. 350.E06 581.1976E06 816.0976E06 ;
   CTRAC1 = EVOL MANU EPS PROGEPS1 SIG PROGSIG1 ;
*
* module tangent valant 491MPa
*-----------------------------
  PROGEPS2 = PROG 0. (350./74020.) 0.3 0.6 ;
  PROGSIG2 = PROG 0. 350.E06 494.9783E06 642.2783E06 ;
   CTRAC2 = EVOL MANU EPS PROGEPS2 SIG PROGSIG2 ;
*
* module tangent valant 199MPa
```

```
*-----------------------------
  PROGEPS3 = PROG 0. (350./74020.) 0.3 0.6 ;
  PROGSIG3 = PROG 0. 350.E06 408.7590E06 468.4590E06 ;
   CTRAC3 = EVOL MANU EPS PROGEPS3 SIG PROGSIG3 ;
*
* module tangent variable
*------------------------
CTRAC = NUAGE 'T'*'FLOTTANT' 'TRAC'*'EVOLUTION'
                  0. CTRAC1 100. CTRAC2 200. CTRAC3  ;
*
* coefficient de dilatation thermique vaiable
*--------------------------------------------
*
  PROGTEMP = PROG 0. 100. 200. ;
  PROGALFA = PROG 0. .25E-3 .5E-3 ;
  EVALPH = EVOL MANU 'T' PROGTEMP 'ALPH' PROGALFA ;
*
   MATvar = MATR MO YOUN 74020.E06 NU 0.3 RHO 7800.
   ALPH EVALPH TRAC CTRAC EPSD 0. DC 0.23 EPSR 0.25 ;
*
************** DEFINITION DU CHARGEMENT ***************
*
   LI1 = PROG 0. 1 31. ; LI2 = PROG 0. 0. 30. ;
   EV = EVOL MANU T LI1 DEP(T) LI2 ;
   CHA1 = CHAR 'DIMP' DEP EV ;
 TEMPREF = MANU CHPO EPROU 1 'T' 30.    ;
   TEMP0 = MANU CHPO EPROU 1 'T' 50.    ;
   TEMP200 = MANU CHPO EPROU 1 'T' 200. ;
   t0 = 0. ; T1 = 1. ; t20 = 20.        ;
   TEMP = TABLE; TEMPERA = TABLE;
   TEMP.0 = t0 ; TEMP.1 = t1 ; TEMP.2 = t20 ;
   TEMPERA.0 = TEMPREF;TEMPERA.1 = TEMP0;            *
   TEMPERA.2 = TEMP200;
   CHA2 = CHAR 'T' TEMP TEMPERA;
   CHA = CHA1 ET CHA2;
*
* LISTE DES PAS DE CALCUL :
*               1.035 MM PREMIER INCREMENT PLASTIQUE
*
si complet ;
  LIS = PROG 1.035 1.1 pas 0.1 7;
sinon;
LIS =   PROG 1.035 1.1 pas 0.1 1.6;
finsi;

*
TAB = TABLE ;
TAB.'PRECISION' = 1E-3;
TAB.'BLOCAGES_MECANIQUES' = CL;
TAB.'CARACTERISTIQUES' = MATvar;
TAB.'CHARGEMENT' = CHA;
TAB.'TEMPS_CALCULES' = LIS;
TAB.'MODELE' = MO;
TAB.'TALPHA_REFERENCE' = 10.      ;
TAB.'TEMPERATURE_REFERENCE' =  TEMPREF ;
PASAPAS TAB ;
*
*-----------------------------------------------------
*  CONTROLE DES RESULTATS AVEC DE LA SOLUTION
*     DE REFERENCE OBTENUE PAR ALGORITHME
*-----------------------------------------------------
*
si complet;
 TA = 4.; TB = 7.;
 REF_D = PROG 7.5E-2 17.7E-2      ;
 REF_P = PROG 10.3E-2 20.9E-2     ;
 REF_S = (PROG -376.9 -369.1)*1E6 ;
SINON;
```

```
 TA = 1.2;  TB = 1.6;
 REF_D = PROG 3.52404E-03     1.24384E-02      ;
 REF_P = PROG 5.75205E-03     1.97372E-02   ;
 REF_S = PROG    -3.52473E+08    -3.57838E+08;
finsi;
*
D1=EXTR (PECHE TAB VARIABLES_INTERNES TA) 'VHWD' 1 1 1;
D2=EXTR (PECHE TAB VARIABLES_INTERNES TB) 'VHWD' 1 1 1;
*
P1=EXTR (PECHE TAB VARIABLES_INTERNES TA) 'EPSE' 1 1 1;
P2=EXTR (PECHE TAB VARIABLES_INTERNES TB) 'EPSE' 1 1 1;
*
S1 = EXTR (PECHE TAB CONTRAINTES TA) 'SMZZ' 1 1 1;
S2 = EXTR (PECHE TAB CONTRAINTES TB) 'SMZZ' 1 1 1;
*
EF_D = PROG D1 D2  ;
EF_P = PROG P1 P2  ;
EF_S = PROG S1 S2  ;
*
PROG_ERD = MAXI (ABS ((EF_D  - REF_D) / REF_D)) ;
PROG_ERP = MAXI (ABS ((EF_P  - REF_P) / REF_P)) ;
PROG_ERS = MAXI (ABS ((EF_S  - REF_S) / REF_S)) ;
*
ERR = MAXI (PROG PROG_ERD PROG_ERP PROG_ERS)    ;
*
temps;
SI ( ERR <EG 0.05 );
   ERRE 0;
SINON;
   ERRE 5;
FINSI;
*
FIN;
```

Test endoaxi3 **Comments**

- The NUAGE operator

```
CTRAC = NUAGE T*FLOTTANT TRAC*EVOLUTION
            0. CTRAC1 100. CTRAC2 200. CTRAC3 ;
```

The NUAGE operator is to define an object of 'NUAGE' type (SET). Here a 'NUAGE' is a set of one couples. The unique couple is 0. CTRAC1 100. CTRAC2 200. CTRAC3 and the two components are T and TRAC.
To define tensile curves versus temperature, we should define three EVOLUTIO objects (CTRAC1 CTRAC2 and CTRAC3) which represent a tensile curve for three different temperatures 0. 100. and 300.°C.

5.5 Test ddi Description sheet

Test name ddi

Calculation type DAMAGE MODEL FOR VISCOPLASTIC MATERIAL

Finite element type QUA8

Topic Two inelastic strains model
Tensile test on a damage viscoplastic material

The structure is an embedded cylindrical test piece subjected to a tensile stress (imposed displacements increasing monotonously). The structure obeys a law according to a damage viscoplastic model. **Goal** Test

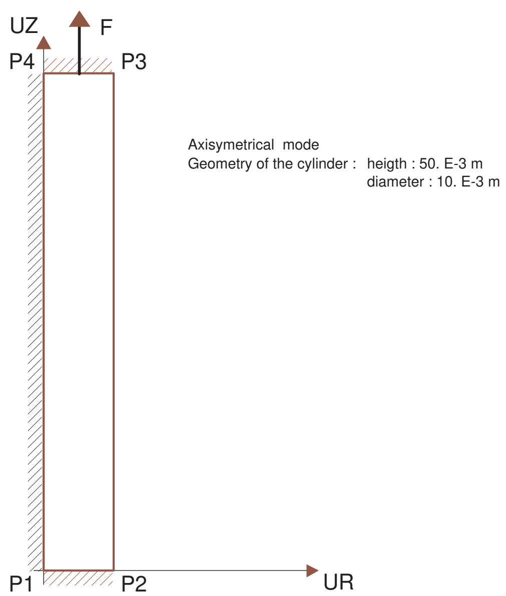
the two inelastic strain model for MODE AXIS and compare Castem's results with the analytical solution.

Version 97' customer version

Model description

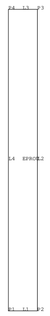
Zirconium alpha 200C

E	=	$8.2\ 10^{10}$
NU	=	0.36

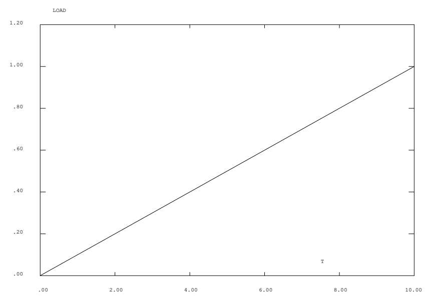


Test ddi **Results**

CASTEM FIGURES



DAMAGE ELASTOPLASTIC MODEL



DAMAGE VISCOPLASTIC MODEL

```
*            Test Ddi.dgibi: Jeux de données        *
*            ------------------------------         *
*                                                   *
OPTI ECHO 0 DIME 2 ELEM QUA8 MODE AXIS;
*
*****************************************************
*               TEST DE VALIDATION                  *
*               ------------------                  *
*       MATERIAU VISCO-PLASTIQUE ENDOMMAGEABLE      *
*    MODELE A DEUX DEFORMATIONS INELASTIQUES (DDI)  *
*                                                   *
*  MAILLAGE:                                        *
*     EPROUVETTE CYLINDRIQUE (AXISYMETRIQUE)        *
*                                                   *
*  CHARGEMENT:                                      *
*     DEPLACEMENT IMPOSE MONOTONE CROISSANT         *
*****************************************************
*
*
   P1 = 0 0;          P2 = 5E-3 0;
   P3 = 5E-3 50E-3;   P4 = 0 50E-3;
*
   L1 = P1 P2 DROIT 1  ;
   L2 = P2 P3 DROIT 1 ;
   L3 = P3 P4 DROIT 1  ;
   L4 = P4 P1 DROIT 1 ;
*
* Maillage
* --------
   EPROU = L1 L2 L3 L4 DALLER PLAN ;
*
* Conditions aux limites
* ----------------------
   CL1 = BLOQ L1 UZ ;
   CL2 = BLOQ L3 UZ;
   CL3 = BLOQ L4 UR ;
   CL = CL1 ET CL2 ET CL3 ;
*
* Modele de calcul
* ----------------
mo = 'MODE' eprou 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
       'VISCOPLASTIQUE' 'DDI' ;
*
* Materiau: Zirconium alpha 200C
* ------------------------------
ma = 'MATE' mo 'YOUN' 8.2D10 'NU' 0.36
       'RP0' 135.D6 'QP' -60.D6 'BP' 120
       'CP1' 34000.D6 'DP1' 250.
       'CP2' 60000.D6 'DP2' 3000. 'KS' 960.D6 'N' 3.4
       'RV0' 70.D6 'QV' -20.D6 'BV' 10.
       'CV1' 24000.D6 'DV1' 300.
       'CV2' 9000.D6 'DV2' 3000. 'CVP1' 0. 'CVP2' 0.;
*
* Temps
* -----
   EPSMAX=0.01;
   EPSPT=0.001;
   NCALC=20;
   TFIN=EPSMAX/EPSPT;
   DT0=TFIN/NCALC;
*
* Chargement
* ----------
   DEP0=DEPI CL2 (EPSMAX*0.05);
   LI1 = PROG 0. tfin ;
   LI2 = PROG 0. 1. ;
   EV = EVOL MANU T LI1 LOAD LI2 ;
   CHA = CHAR 'DIMP' DEP0 EV ;
*
* Calcul
* ------
   TAB = TABLE ;
   TAB.'BLOCAGES_MECANIQUES' = CL;
   TAB.'CARACTERISTIQUES' = MA;
   TAB.'MODELE' = MO;
   TAB.'CHARGEMENT' = CHA;
   LIS=PROG 0. pas dt0 tfin;
   TAB.'TEMPS_CALCULES' = LIS;
   PASAPAS TAB ;
*
*  Comparaison avec les valeurs de reference
*  -----------------------------------------
  dim0 = 'DIME' ( tab .'TEMPS' ) - 1;
  var0 = tab .'VARIABLES_INTERNES'. dim0 ;
  sigm0= tab .'CONTRAINTES'.dim0;
  p0 =  MAXI (EXCO var0 'VIPW');
  v0 =  MAXI ( EXCO var0 'V');
  sig0 = MAXI ('EXCO' sigm0 'SMZZ');
*
  p_th = 4.17896E-03;
  v_th = 3.14432E-03;
  sig_th = 2.19491E+08;
  errp = 100. * (ABS((p_th - p0)/p_th));
  errv = 100. * (ABS((v_th - v0)/v_th));
  errsig = 100. * (ABS((sig_th - sig0)/sig_th));
*
  SI ( (errp > 0.1) OU (errv > 0.1) OU (errsig > 0.1));
    ERRE 5 ;
  SINON;
    ERRE 0 ;
  FINSI;
*
FIN;
```

Test ddi **Comments**

- Two inelastic strain viscoplastic model (DDI)

```
MO = MODE EPROU MECANIQUE ELASTIQUE ISOTROPE
         VISCOPLASTIQUE DDI ;
MA = MATE MO YOUN 8.2D10 NU 0.36
   'RP0' 135.D6 'QP' -60.D6 'BP' 120
   'CP1' 34000.D6 'DP1' 250.
   'CP2' 60000.D6 'DP2' 3000. 'KS' 960.D6 'N' 3.4
   'RV0' 70.D6 'QV' -20.D6 'BV' 10.
   'CV1' 24000.D6 'DV1' 300.
   'CV2' 9000.D6 'DV2' 3000. 'CVP1' 0. 'CVP2' 0.;
```

The equations for the two inelastic strain model are as follows:

-> Notation:

	S	stress tensor
	EP	plastic strain tensor
	p	accumulated equivalent plastic strain
	EV	creep strain tensor
	v	accumulated equivalent creep strain
	Xpi, Xvi	kinematic stress hardening variables (i=1,2)
	Rp, Rv	isotropic stress hardening variables
	J2	deviatoric tensor second invariant
	np	normal from Fp criterion
	nv	normal from Fv criterion
	$< a >$	positive part of a

Xp = Xp1
Xv = Xv1 for a single centre
Xp = Xp1+Xp2
Xv = Xv1+Xv2 for two centres

-> Criteria:
Fp = J2(S-Xp) - Rp
Fv = J2(S-Xv) - Rv

-> Stres shardening:
Xp1 = CP1*ALPHAp1 + CVP1*ALPHAv1
Xp2 = CP2*ALPHAp2 + CVP2*ALPHAv2
Xv1 = CV1*ALPHAv1 + CVP1*ALPHAp1
Xv2 = CV2*ALPHAv2 + CVP2*ALPHAp2
dALPHApi = dEp - DPi*dp*np (i=1,2)
dALPHAvi = dEv - DVi*dv*nv (i=1,2)
Rp = RP0 + QP*(1-exp(-BP*p))
Rv = RV0 + Qv*(1-exp(-BV*v))

-> Flow rule:
dp verifies dFp=0
dEp = 3/2 * dp * < S-Xp > / J2(S-Xp)
dV = (< Fv > / KS) ** n
dEv = 3/2 * dv * < S-Xv > / J2(S-Xv)

Data to be input:

Evolution law for the centres Xpi and Xvi :
'CP1' : ALPHAp1 coefficient
'CP2' : ALPHAp2 coefficient
'CV1' : ALPHAv1 coefficient
'CV2' : ALPHAv2 coefficient
'CVP1' : Visco-plastic coupling coefficient
'CVP2' : Visco-plastic coupling coefficient

Evolution law for the ALPHApi et ALPHAvi internal variables:
'DP1' : ALPHAp1*dp coefficient
'DP2' : ALPHAp2*dp coefficient
'DV1' : ALPHAv1*dv coefficient
'DV2' : ALPHAv2*dv coefficient

Evolution law for isotropic stress hardening:
'BP' : P coefficient
'QP' : plastic strain isotropic hardening coefficient
'RP0' : initial plastic threshold value
'BV' : V coefficient
'QV' : creep strain isotropic hardening coefficient
'RV0' : initial creep threshold value

Evolution law for the creep strain:
'KS' : creep threshold normalization coefficient
'N' : creep law exponent

Chapter 6

FRACTURE MECHANICS

6.1 Test rupt1 Description sheet

Test name rupt1

Calculation type MECHANICS ELASTIC-PLASTIC AXI

Finite element type QUA8

Topic
STRESS INTENSITY FACTOR IN LINEAR MECHANICS ELASTIC OF A SOLID CYLINDER WITH A CIRCUMFERENCIAL CRACK

The structure is a solid cylinder with a circumferencial crack subjected to tensile stress. The goal is to calculate the Crack Opening Displacement (COD) at the crack tip and the stress factor in linear elastic mechanics of this structure. The theoreticals values are given by the relationships :

$K = \sigma_t\sqrt{\pi a}$

$\sigma_t = p/2\pi R_m t$ $\quad R_m$ *is the mean radius*

$F = 1.1 + A(1.948(a/t)^{0.5} + 0.3342(a/t)^{4.2})$

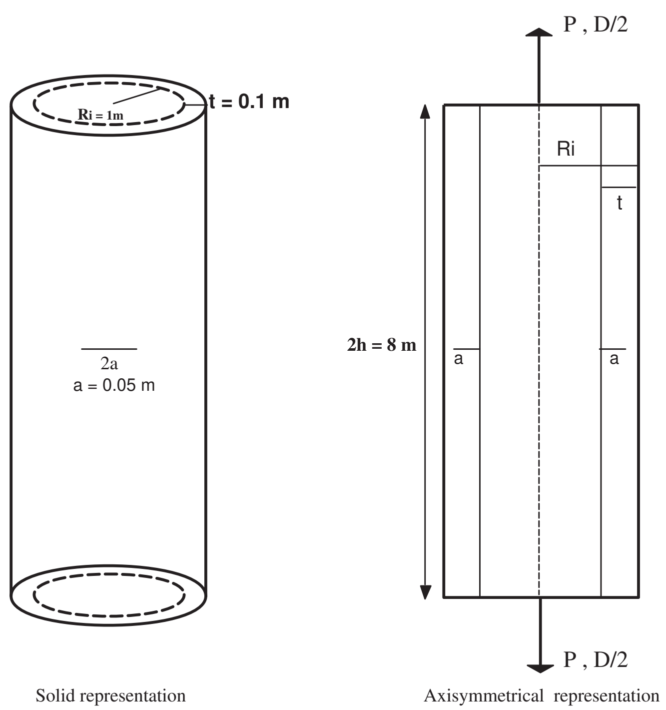
$A = (0.125(R_i/t) - 0.25)^{0.25})$

For this problem the CASTEM solution for the stress factor K is given by the relation ship :

$K = (\sqrt{EG/1-\nu^2})$ $\quad G$ *is the energy release rate*

Reference CASTEM Test NAFEMS : Ductile Fracture Handbook, A. ZAHOOR, EPRI, 1990 : rupt1 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97' customer version **Model**

description



$$
\begin{aligned}
\text{E} &= 2.1\ 10^{11}\ Pa \\
\text{NU} &= 0.3 \\
\text{a} &= 0.05 m
\end{aligned}
$$

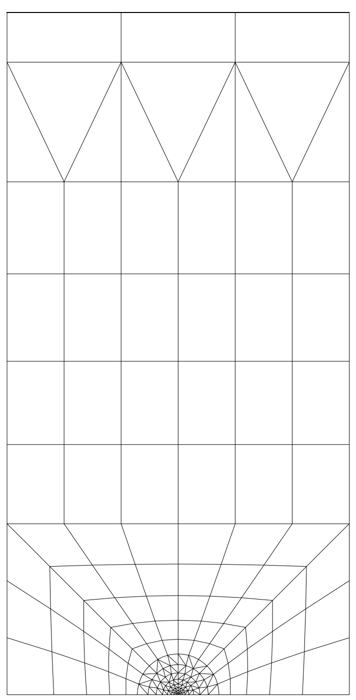
Test rupt1 **Results**

RESULTS

	$K1(MPam^{1/2})$
Reference	1.0848
Castem	1.0855

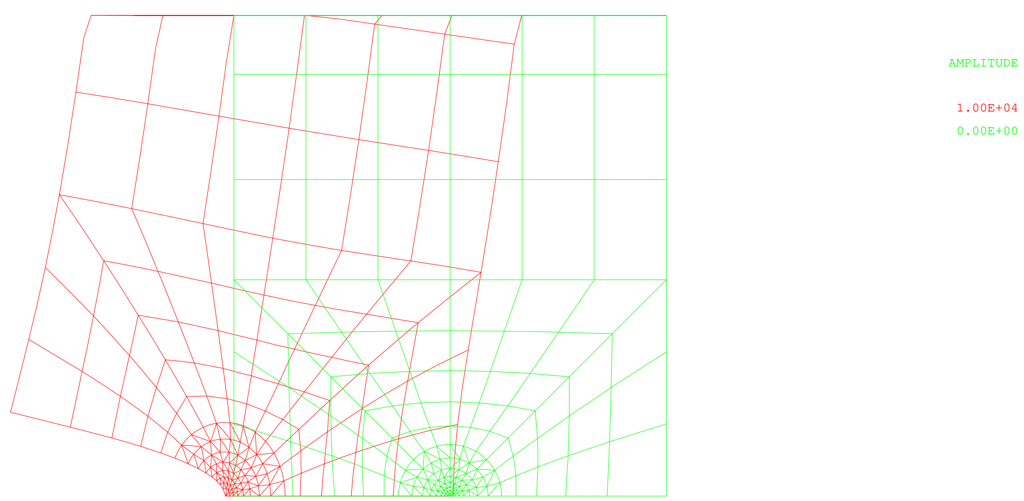
CASTEM FIGURES

MESH OF THE CYLINDER



MESH OF THE CYLINDER

deformation of the cylinder subjected to tensile stress



deformation of the cylinder subjected to tensile stress

```
*          Test Rupt1.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0;
GRAPH = 'N';
SAUT PAGE;
*****************************************************
*
*             QUALIFICATION DU CALCUL DE K EN
*         ELASTICITE LINEAIRE SUR UN CYLINDRE AVEC
*         UNE FISSURE DEBOUCHANTE CIRCONFERENTIELLE
*
* Le calcul est compare a celui obtenu par A ZAHOOR
*****************************************************
* R = rayon interne de la tuyauterie (M)
* B = épaisseur de la tuyauterie (M)
* L = longueur de la tuyauterie (M)
* A = longueur de la fissure (M)
* MYOU = module d'Young (Pa)
* TA = chargement appliqué (N)
***
TA = 1.E6;
R = 1.;
B = 0.1;
A = B / 2.;
L = R * 4.;
MYOU = 2.0E11;
***
*** Maillage
***
OPTION DIME 2 ELEM QUA8 MODE AXIS;
t = a / 100.; densite t ; pf = (a 0.);
c1 = (c ( pf moin (t 0.)) pf ( pf plus (0. t)))
      c pf ( pf plus (t 0.));
sf = cout pf c1;
r1 = t ; rr1 = t;
repeter bhomo 7;
   ri = r1 + ( 0.3 * r1 );
   rri = rr1 + ri; dens ri;
   ci = (c (pf moin (rri 0.)) pf ( pf plus (0. rri)))
         c pf (pf plus (rri 0.));
   sf = sf et (cout c1 ci);
   c1 = ci ; r1 = ri ; rr1 = rri;
fin bhomo;
dens (a / 3.);
p0  = (0. 0.) ; p1 = (b 0.);
p2  = p0 plus (0. a) ; p3 = p1 plus (0. a);
pi1 = ci poin 1 ; l1 = pi1 d p0 ; n = (nbel l1) * -1;
pi2 = ci poin 4 ; l2 = pi2 d n p2;
pi3 = ci poin 10 ; l3 = pi3 d n p3;
pi4 = ci poin 13 ; l4 = pi4 d p1;
ci  = inve ci ; ligh = p2 d p3;
sc1 = dall l1 (p0 d p2) (inve l2) (ci comp pi2 pi1);
sc2 = dall l2 ligh (inve l3) (ci comp pi3 pi2);
sc3 = dall l3 (p3 d p1) (inve l4) (ci comp pi4 pi3);
sc  = sc1 et sc2 et sc3;
dens (a / 2.);
mrest1 = ligh tran (0. (2.*a)) dini (40*t) dfin (50*t);
l1 = mrest1 cote 3;
YY1 = coor 2 (point l1 init);
l2 = D 3 (B (YY1 + (0.7*a))) (0. (yy1 + (0.7*a)));
S1 = COUT L1 L2;
YY1 = COOR 2 (l2 point init);
mrest2 = l2 tran (0. (L - YY1)) dini (80*t)
                                dfin (150*t);
sut = sf et sc et mrest1 et S1 et mrest2;
ELIM 1.E-8 SUT;
DEPL PLUS SUT (R 0.);
* L1 = (CONT SUT) ELEM APPU (SUT POIN
*      DROI (R 0.) ((B + R) 0.) 1.E-8);
* L2 = (CONT SUT) ELEM APPU (SUT POIN
       DROI ((B + R) 0.) ((B + R) L) 1.E-8);
L3 = (CONT SUT) ELEM APPU (SUT POIN
     DROI ((B + R) L) (R L) 1.E-8);
L4 = (CONT SUT) ELEM APPU (SUT POIN
     DROI (R L) (R 0.) 1.E-8);
L5 = (CONT SUT) COMP P1 PF;
SI ( NEG GRAPH 'N' );
   TITR 'MAILLAGE DU CYLINDRE';
   TRAC sut;
FINSI;
******
****** RESOLUTION EN ELASTICITE LINEAIRE
******
MO1 = MODELE SUT MECANIQUE ELASTIQUE
      PLASTIQUE ISOTROPE;
MA1 = MATER MO1 YOUNG MYOU NU 0.3 TRAC COUTRAC;
RI = (BLOQ UZ L5) ET (RIGI MA1 MO1);
AIR1 = PI*(((R + B)**2.) - (R*R));
FOR1 = PRES 'MASS' MO1 (0. - (TA/AIR1)) L3;
DEP1 = RESO FOR1 RI;
SIG1 = SIGMA MO1 MA1 DEP1;
SI ( NEG GRAPH 'N' );
 TITR 'DEFORMATION DU CYLINDRE SOUS TRACTION UNIFORME';
 TRAC (DEFO SUT DEP1);
FINSI;
***
*** Solution COD (MM) et K (MPA M^0.5) de castem
***
COD_CAL = (EXTR DEP1 'UZ' P1)*2000.;
SUPTAB = TABLE ;
SUPTAB.'OBJECTIF' = MOT 'J';
SUPTAB.'LEVRE_SUPERIEURE' = l1 diff l5;;
SUPTAB.'FRONT_FISSURE' = PF ;
SUPTAB.'MODELE' = MO1;
SUPTAB.'CARACTERISTIQUES' = ma1;
SUPTAB.'SOLUTION_RESO' = dep1;
SUPTAB.'CHARGEMENTS_MECANIQUES' = for1;
SUPTAB.'COUCHE' = 5;
G_THETA SUPTAB;
K_CAL = (MYOU*(SUPTAB.'RESULTATS')/(1 - (0.3**2)))**0.5;
K_CAL = K_CAL*1.E-6;
***
*** Solution Analytique Zahoor
***
RAP1 = R / B;
SI (RAP1 < 10.); GRANDA = ((0.125*RAP1) - 0.25)**0.25;
FINSI;
SI (RAP1 >EG 10.); GRANDA = ((0.4*RAP1) - 3.00)**0.25;
FINSI;
F = (1.9480*((A/B)**1.5)) + (0.3342*((A/B)**4.2));
F = 1.1 + (GRANDA*F);
SIGT = TA / AIR1;
K_ZAH = SIGT*((PI*A)**0.5)*F;
K_ZAH = K_ZAH*1.E-6;
ERR1 = abs ((K_CAL - K_ZAH)/K_ZAH);
MESS '  Facteur K CASTEM =' K_CAL '(Mpa.M^0.5)';
MESS '  Facteur K ZAHOOR =' K_ZAH '(Mpa.M^0.5)';
MESS '  Erreur relative =' ERR1;
SI (ERR1 <  1.E-2);
    ERRE 0;
SINO;
    ERRE 5;
FINSI;
FIN;
```

Test rupt1 **Comments**

1. GEOMETRY AND MESHING (OPERATOR COUTURE)
   To account for singular displacement and stress distribution at the crack tip the mesh must be finer at the crack tip. Here the mesh is obtain with the COUT operator:
   For example:

```
t = a / 100.; densite t; pf = (a 0.);
c1 = (c ( pf moin (t 0.)) pf ( pf plus (0. t)))
      c pf ( pf plus (t 0.));
trac c1 ;
sf = cout pf c1;
```

   Thus, the COUT operator constructs a surface which connects two lines or a line and point by means of triangles. here pf is a point (POINT type) to be connected to the line c1 (line between which the surface is generated (MAILLAGE type).

2. RESOLUTION IN ELASTIC MECHANICS (G_THETA PROCEDURE)

```
SUPTAB = TABLE ;                           SUPTAB.'OBJECTIF' = MOT 'J';
SUPTAB.'LEVRE_SUPERIEURE' = l1 diff l5;    SUPTAB.'FRONT_FISSURE' = PF ;
SUPTAB.'MODELE' = MO1;                     SUPTAB.'CARACTERISTIQUES' = ma1;
SUPTAB.'SOLUTION_RESO' = dep1;             SUPTAB.'CHARGEMENTS_MECANIQUES' = for1;
SUPTAB.'COUCHE' = 5;                       G_THETA SUPTAB;
K_CAL = (MYOU*(SUPTAB.'RESULTATS')/(1 - (0.3**2)))**0.5;
K_CAL = K_CAL*1.E-6;
```

   The G_THETA procedure has two objectives :

   (a) calculate line integrals of fracture mechanics as follows :
      - the J line integral of an isotropic material, characterizing the crack tip fields in elasto-plasticity. In 3D massive element case, composite materials are not yet authorised.
      - the J line dynamic integral of an isotropic material, characterizing the crack tip fields for elasto-dynamic problems. In 3D massive element case, composite materials are not yet authorised.
      - the C* line integral of an isotropic material, characterizing the crack tip fields in the case of secondary creep problems. Applied loadings must be mechanical and composite materials are not yet authorised in 2D or 3D configurations.
      - the C*(h) line integral of an isotropic material, characterizing the crack tip fields in the case of primary or tertiary creep problems. Applied loadings must be mechanics and composite materials are not yet acceptable in 2D configurations, nor in 3D configurations.
      - the derivative dJ/da (a : crack length) line integral of an isotropic homogeneous material, required in the stability analysis of a single crack or a system of interacting cracks. Only the massive elements (2D or 3D) are allowed and composite materials are not yet available for calculating this derivative integral.

   (b) separate the stress intensity factors K1, K2 (and K3 in 3D) for elastic problems in 2D, 3D (massive elements only) or axisymetrical configurations. For separating a mixed mode, the current version of G_THETA procedure allows only isotropic homogeneous materials.

For this probleme the goal is to compute the parametr $G = J$ in elasticity witch representsis the energy release rate.
Therefore : Input data of the procedure is a TABLE object type, SUPTAB, wich indexes are objects of MOT type, written in capital and described as follows :

- SUPTAB.'OBJECTIF' = MOT type, specifying the calculation aim, equal to 'J' for calculating the J line integral, characteristics in elasto-plasticity.
- SUPTAB.'LEVRE_SUPERIEURE' = following the definition convention, this object (MAILLAGE type) gives the crack's superior lip.
- SUPTAB.'FRONT_FISSURE' = POINT type in 2D or 3D shell element case, MAILLAGE type in 3D massive element case. This object gives the tip of the crack.
- SUPTAB.'MODELE' = modele object (type MMODEL) covering the entire structure
- SUPTAB.'CARACTERISTIQUES' = MCHAML object of CARACTERISTIQUES subtype In the case of calculations with shell elements, this object includes material's properties (Young's modulus, Poisson's coefficient ...) and geometrical parameters like shell thickness, generated by MATE or CARA operator.
- SUPTAB.'SOLUTION_RESO' = displacement solution (CHPOINT type) resulting from RESO operator.
- SUPTAB.'CHARGEMENTS_MECANIQUES' = all external mechanical loadings (CHPOINT type) if they exist.
- SUPTAB.'COUCHE' = ENTIER type. It is the number of layers of elements surrounding the crack tip in 2D (or surrounding a point on the crack front in 3D), that support the virtual crack extension. If COUCHE = 0, the THETA field is equal to 1 only at the crack tip (in 2D), or to 1 for all points on the crack front (in 3D). In general, the bigger the number of layers of elements , the accurate more the value calculated by the present G_THETA procedure. But the elements included in the number of layers should not reach the mesh border.

The CASTEM solution for the stress factor K is geven by the relation ship :

$$K = (\sqrt{EG/1-\nu^2}) :$$

```
K_CAL = (MYOU*(SUPTAB.'RESULTATS')/(1 - (0.3**2)))**0.5;
K_CAL = K_CAL*1.E-6;
```

6.2 Test rupt2 Description sheet

Test name rupt2

Calculation type ELASTIC MECHANICS

Finite element type QUA8 PLANE STRAIN MODE

Topic THE USE OF G-INTEGRAL IN THERMAL STRESS FOR AN EDGE CRACKED STRIP

The structure is a solid edge cracked strip which ends are constrained. It is subject to a linear temperature gradient through the thickness with zero temperature at mid-thickness and temperature T_0 at the right edge $(x = w/2)$.
Due to symmetry only one half of the strip is analysed. The CASTEM solution for the stress intensity factor K is given by the relationship :

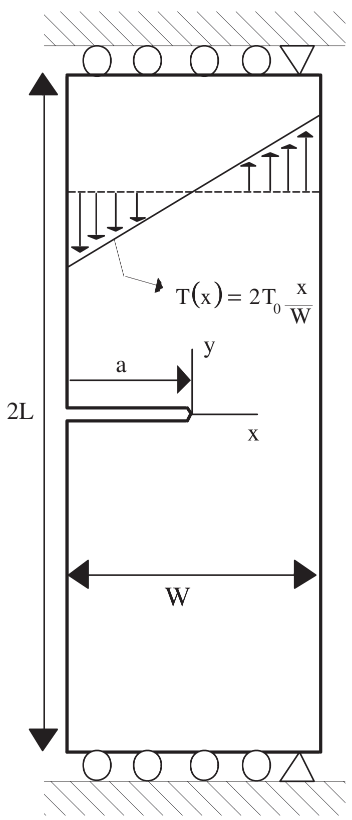
$K = (\sqrt{EG/1-\nu^2})$ $\quad$ *G is the energy release rate*

Finally the K values for 7 differente temperatures $\theta(\theta 1, \theta 2, \theta 3 \ldots \theta 7)$ are computed and compared with the results obtained with WILSON and YU relationship :

$K = (E\alpha T_0/1-\nu)F\sqrt{(\Pi a)}$ $\quad$ $F = 0.514$

Reference CASTEM Test NAFEMS : The use of J-integral in thermal stress crack problems
International Journal of Fracture :
rupt2 Modélisation des structures élastiques dans CASTEM 2000.

Version 97' customer version **Model description**



Edge cracked strip subject to thermal loading

GEOMERTY :
Width of the edge : W = 200 mm
Length of the edge : 2L = 800 mm
Length of the crack : a = 100 mm

MATERIAL CHARACTERISTICS :
E = 20000 daN mm^(-2)
alpha = 5.10E-6 °C^(-1)
nu = 0.3

LOADING AND BOUNDARY CONDITIONS :
1- Linear Temperature gradient with zero temperature at mid thickness and temperature T_0 at the right edge
$T(x) = 2T_0\ x/W\ [\,-w/2 < x < w/2]$
$T_0 = 100°C$

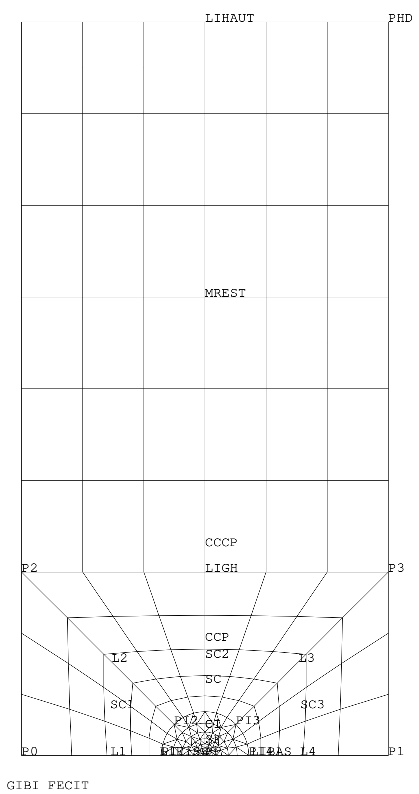
2- The vertical displacement (Uy) of the upper and lower extremity is blocked

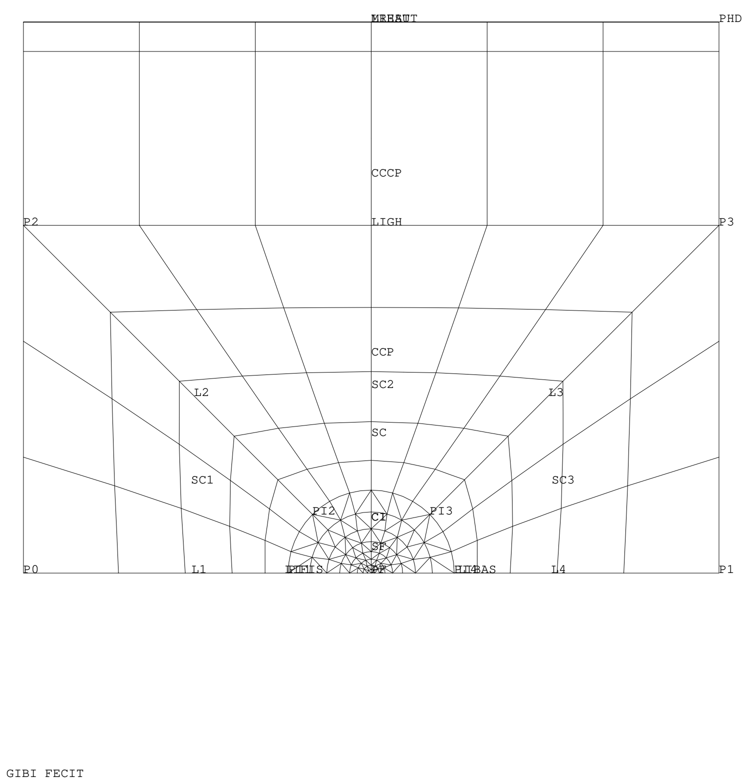
Test rupt2 **Results**

RESULTS

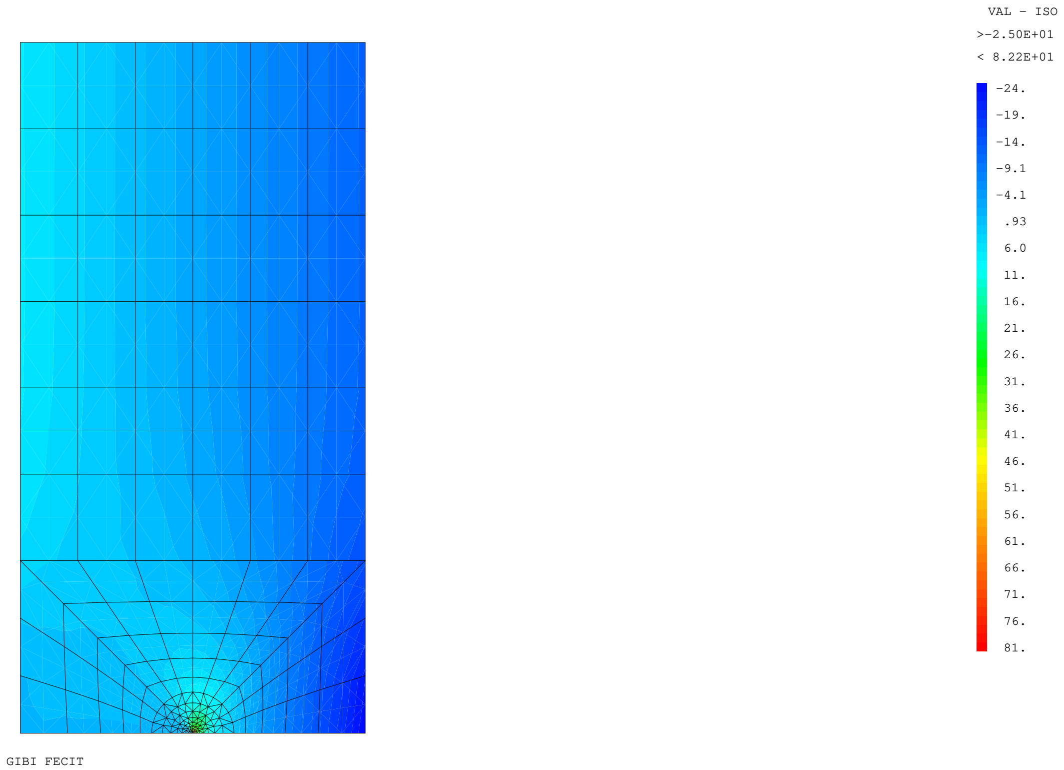
	$K1(daNmm^{-3/2})$
Reference	130.15
Castem	125.88

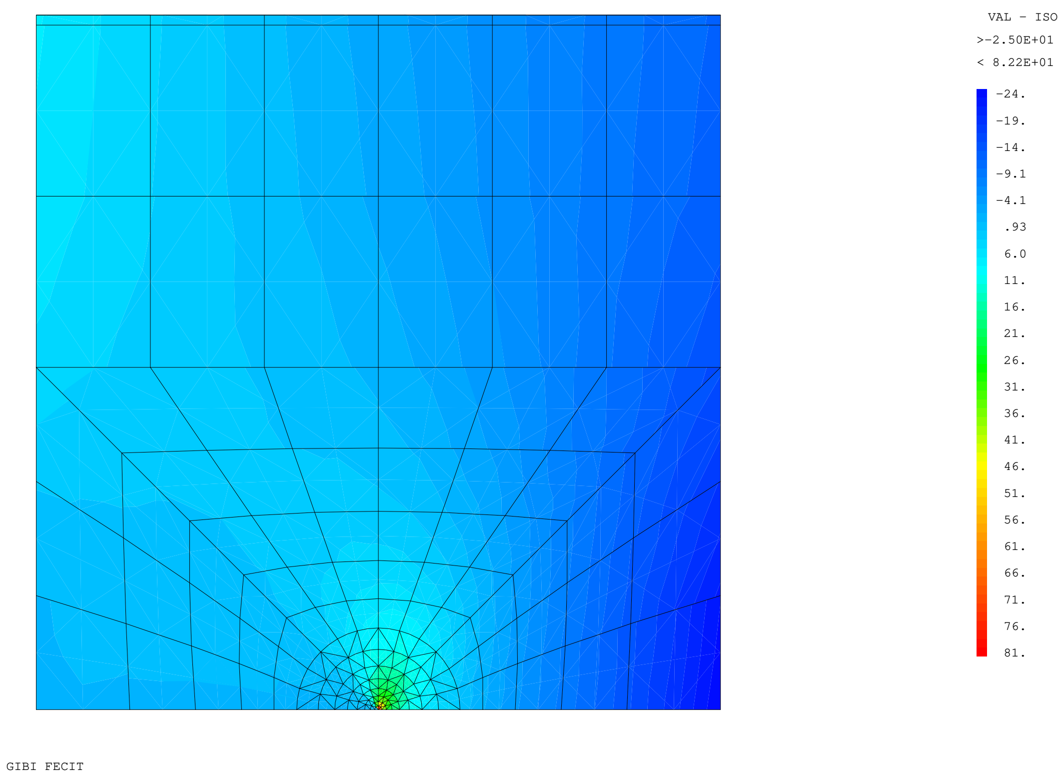
CASTEM FIGURES





Test rupt2 **Results**





```
*          Test Rupt2.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0                                       ;
GRAPH = 'N'                                         ;
SAUT PAGE                                           ;
*                                                   ;
****************************************************;
*                                                   ;
*                                                   ;
*             QUALIFICATION DU CALCUL DE G          ;
*             EN THERMO-ELASTICITE LINEAIRE         ;
*             SUR UNE PLAQUE A FISSURE LATERALE     ;
*             EVALUATION DU FACTEUR DE FORME        ;
*                                                   ;
*                                                   ;
* le calcul est compare a celui obtenu par          ;
* WILSON et YU                                      ;
* dans The use of J-Integral in thermal stress crack ;
* problems international Journal of Fracture (1979)  ;
*                                                   ;
****************************************************;
*                                                   ;
OPTION DIME 2 ELEM QUA8 MODE PLAN DEFO              ;
*                                                   ;
*---------------- DEFINITION DU MAILLAGE -----------;
*                                                   ;
a = 100. ; b = 200. ; h = 400. ;t = a / 100.        ;
densite  t ; pf = (a 0.)                            ;
c1 = (c ( pf moin (t 0.)) pf ( pf plus (0. t)))
      c pf ( pf plus (t 0.))                        ;
sf = cout pf c1;
r1 = t ; rr1 = t                                    ;
repeter bhomo 7;
   ri  =  r1 + ( 0.3 * r1 )                         ;
   rri = rr1 + ri                                   ;
   dens ri                                          ;
   ci = ( c ( pf moin (rri 0.)) pf ( pf plus (0. rri)))
          c pf ( pf plus (rri 0.))                  ;
   sf = sf et ( cout c1 ci )                        ;
   c1 = ci ; r1 = ri ; rr1 = rri                    ;
fin bhomo                                           ;
dens (a / 3.)                                       ;
p0  = (0. 0.) ; p1 = (b 0.)                         ;
p2  = p0 plus (0. a) ; p3 = p1 plus (0. a)          ;
pi1 = ci poin 1  ; l1  = pi1 d p0                   ;
n = (nbel l1) * -1                                  ;
pi2 = ci poin 4  ; l2  = pi2 d n p2                 ;
pi3 = ci poin 10 ; l3  = pi3 d n p3                 ;
pi4 = ci poin 13 ; l4  = pi4 d   p1                 ;
ci  = inve ci    ;ligh = p2  d   p3                 ;
sc1 = dall l1 (p0 d p2) (inve l2) (ci comp pi2 pi1)  ;
sc2 = dall l2 ligh (inve l3) (ci comp pi3 pi2)      ;
sc3 = dall l3 (p3 d p1) (inve l4) (ci comp pi4 pi3)  ;
sc  = sc1 et sc2 et sc3                             ;
dens (a / 2.)                                       ;
mrest  = ligh tran (0. (h - a)) dini (a / 2.)       ;
lihaut = mrest cote 3                               ;
phd    = lihaut poin init                           ;
ccp    = sf et sc et mrest                          ;
elim  ccp  0.001                                    ;
cccp   = cont ccp                                   ;
lifis  = cccp comp pf p0                            ;
libas  = cccp comp p1 pf                            ;
*                                                   ;
*------------------ CREATION DU MODELE -------------;
*                                                   ;
objaf = modl ccp mecanique elastique isotrope       ;
```

```
*                                                   ;
*---------- DEFINITION DES CARACTERISTIQUES ----------;
*----------   MATERIELLES ET GEOMETRIQUES   ----------;
*                                                   ;
mat = matr objaf  YOUN 2.e4  NU 0.3 ALPH 5.e-6      ;
*                                                   ;
*---------- CALCUL DES RIGIDITES ELEMENTAIRES --------;
*---------     ET DEFINITION DES BLOCAGES    ---------;
*                                                   ;
rig  = rigi objaf mat                               ;
cdl1 = bloq Uy libas                                ;
cdl2 = bloq Uy lihaut                               ;
cdl3 = bloq Ux phd                                  ;
*                                                   ;
*-------- DEFINITION DU CHAMP DE TEMPERATURES --------;
*                                                   ;
chx = coor 1 ccp                                    ;
cha = manu chpo ccp 1 scal a                        ;
chx = nomc 'T' (chx - cha)                          ;
cht = chx * (100. / a)                              ;
ch0 = 0 * cht                                       ;
sigth = thet MAT objaf cht                          ;

*                                                   ;
*-------- RESOLUTION ET CALCUL DES CONTRAINTES -------;
*                                                   ;
U    = reso (rig et cdl1 et cdl2 et cdl3 )
       (bsig objaf sigth)   ;
SIG  = sigm mat objaf U                             ;
SIG  = SIG - sigth                                  ;
sigy = exco SIG smyy                                ;
SI ( NEG GRAPH 'N' )                                ;
     TRAC  CCP                                      ;
     TRAC SIGY OBJAF CCP                            ;
FINSI                                               ;
*                                                   ;
*------ INITIALISATION DE LA TABLE EN ENTREE ---------;
*----       DE LA PROCEDURE G_THETA        ----------;
*                                                   ;
SUPTAB = TABLE ;
SUPTAB.'OBJECTIF' = MOT 'J';
SUPTAB.'LEVRE_SUPERIEURE' = lifis;
SUPTAB.'FRONT_FISSURE' = PF ;
SUPTAB.'MODELE' = objaf;
SUPTAB.'CARACTERISTIQUES' = mat;
SUPTAB.'SOLUTION_RESO' = u;
SUPTAB.'TEMPERATURES' = cht;
SUPTAB.'COUCHE' = 4;
*                                                   ;
*----------- APPEL A LA PROCEDURE G_THETA ------------;
*                                                   ;
SAUT PAGE                                           ;
G_THETA suptab                                      ;
*                                                   ;
*------ RECUPERATION DU RESULTAT ET COMPARAISONS -----;
*                                                   ;
G = suptab.'RESULTATS'                              ;
K = (20000. * G / (1 - (0.3**2)))**0.5              ;
F = K * (1 - 0.3) /
    ( 20000 * 5.e-6 * 100 * ((100*pi)**0.5))        ;
mess 'taux de restitution energetique       G :' G  ;
mess 'coefficient d intensite de contrainte  K :' K  ;
mess 'facteur de forme  calcule          Fcal :' F  ;
mess
'facteur de forme analytique        Fthe :' 0.514   ;
*                   CODE FONCTIONNEMENT             ;
RESI = abs (( F - 0.514 ) / 0.514 )                 ;
SI (RESI <  5E-2)                                   ;
```

```
    ERRE  0                                             ;
SINO                                                    ;
    ERRE  5                                             ;
FINSI                                                   ;
fin;
```

Test rupt2 **Comments**

1. DEFINITION OF THE LINEAR TEMPERATURE GRADIENT

```
chx = coor 1 ccp              ;
cha = manu chpo ccp 1 scal a  ;
chx = nomc 'T' (chx - cha)    ;
cht = chx * (100. / a)        ;
ch0 = 0 * cht                 ;
sigth = thet MAT objaf cht    ;
```

- COOR OPERATOR :
  The COOR operator enables the user to recover the Nith coordinate (for this example it is the first coordinate$N = 1$) of a POINT, MAILLAGE, CHPOINT or MCHAML object type (here CCP is a MAILLAGE (MESH) type).
  For a CCP mesh object type, the operator supplies the coordinates of the nodes supporting the mesh.
- MANU OPERATOR :
  The MANU operator enables the user to simply create objets of the following types : MAILLAGE, CHPOINT, SOLUTION, RIGIDITE. here the MANU operator constructs a field by point by means of the key word 'CHPO'. :
  CCP is the geometrical support (MAILLAGE type)
  1 is the number of components of the field by points (ENTIER type)
  SCAL is a name of components (MOT type) not exceeding 4 characters
  a is a list of the value allocated to each node of CCP.
- NOMC OPERATOR :
  The NOMC operator enables the user either to change the component na of a field by points or to replace a sublist of components by a second one :
  (chx - cha) is CHPO1 field by points (CHPOINT type) and 'T' (Temperature) is a new name allocated to the component (MOT type) . CHX is the generated object (CHPOINT type) .
  NOTE :
  The field by points CHPO1 must be composed of a single component which name is arbitrary.
- THET OPERATORS :
  The THETA operator calculates the stresses associated with a temperature field, i.e. :
  $SIGMA = HOOK * EPSTHER$
  in which HOOK is Hooke's matrix and EPSTHER corresponds to the strains of thermal origin :

  ```
  sigth = thet MAT objaf cht
  ```

  MAT is a field of material and geometrical properties (MCHAML type, CARACTERISTIQUES subtype)
  OBJAF is a model object (MMODEL type)
  CHT is a temperature field (CHPOINT type).
  SIGTH is a stress field (MCHAML type, CONTRAINTES subtype).
  Note :
  For the shell elements, the temperature field must have three components with the following names : TINF ,T, and TSUP which define respectively the temperature on the bottom , mid-surface and top layers. For the other elements, the temperature field must have one component with the following name : T. You may recover the thermal strains from the thermal stresses using the ELAS operator.

2. RESOLUTION G_THETA PROCEDURE
   The CASTEM solution for the stress intensity factors K is given by the G_THETA PROCEDURE from the relationship :

   $$K = (\sqrt{EG/1-\nu^2}) \qquad G \textit{ is the energy release rate}$$

6.3 Test rupt3 Description sheet

Test name rupt3

Calculation type MECHANICS ELASTIC

Finite element type QUA8 PLANE STRAIN MODE

Topic THE USE OF G-INTEGRAL IN TENSILE STRESS TEST FOR A SIMPLE PLATE WITH A CENTRAL CRACK

The structure is a solid plate containing a central crack . This structure is subject to a uniform load by a simple tensile stress test. Due to symmetry only 1/4 of the structure is analysed. The CASTEM solution for the stress intensity factors K is given by the relationship :

$K = (\sqrt{EG/1-\nu^2})$ $\quad G$ *is the energy release rate*

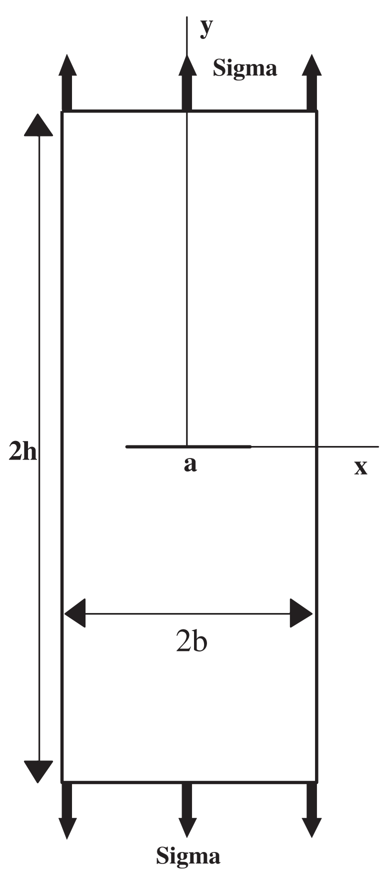
Finally this CASTEM K value for the stress intensity factors is compared with the result obtained with the analytical formula of ROOKE and CARTWRIGHT from the relationship :

$$\frac{K}{K_O} = \frac{1 - 0.5\frac{a}{b} + 0.326(\frac{a}{b})^2}{(1-\frac{a}{b})^{1/2}}$$

$$K_0 = \sigma\sqrt{\pi}a$$

Reference CASTEM Test NAFEMS : D.P.Rooke and D. J. Cartwright in Copendium of Intensity Factors HMSO (1976) : rupt3 Modélisation des structures élastiques dans CASTEM 2000.

Version 97' customer version **Model description**



Center cracked strip subject to Tensile Stress

GEOMERTY :
Width of the edge : 2b = 400 mm
Length of the edge : 2h = 1200 mm
Length of the crack : 2a = 200 mm

MATERIAL CHARACTERISTICS :
E = 20000 daN mm^(-2)
nu = 0.3

LOADING CONDITIONS :
1- Uniform load (Tensile stress)
Sigma = 1 daN mm^(-2)

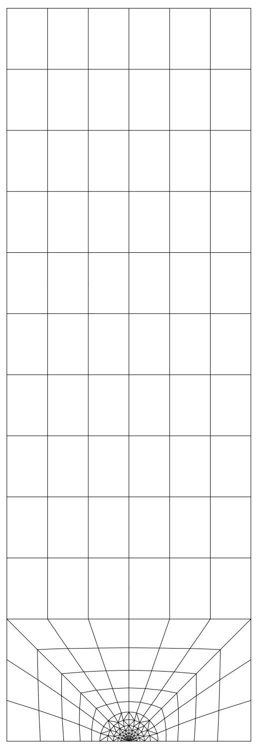
Test rupt3

Results

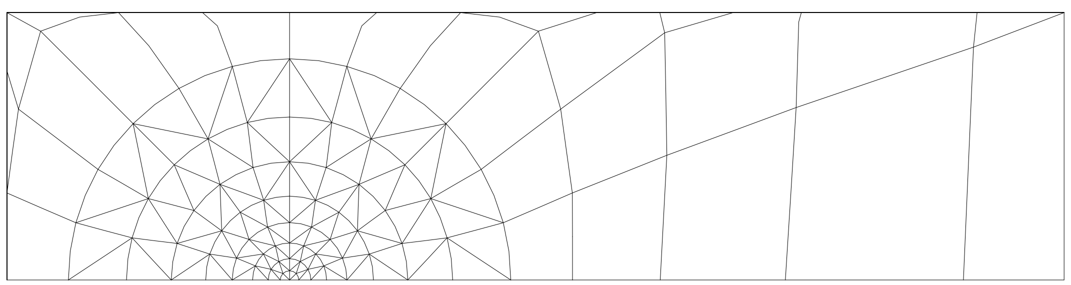
RESULTS

	$K1(daNmm^{-3/2})$
Reference	20.84
Castem	21.02

CASTEM FIGURES



GIBI FECIT



GIBI FECIT

```
*          Test Rupt3.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0                                       ;
GRAPH = 'N'                                         ;
SAUT PAGE                                           ;
*                                                   ;
****************************************************;
*                                                   ;
*                                                   ;
*             QUALIFICATION DU CALCUL DE G          ;
*             EN ELASTICITE LINEAIRE SUR            ;
*             UNE PLAQUE A FISSURE INTERNE          ;
*                                                   ;
*                                                   ;
* le calcul est compare a celui obtenu par          ;
* ROOKE et CARTWRIGHT                               ;
* dans Compendium of Stress Intensity Factors       ;
*                                                   ;
****************************************************;
*                                                   ;
OPTION DIME 2 ELEM QUA8 MODE PLAN DEFO              ;
*                                                   ;
*----------------- DEFINITION DU MAILLAGE ------------;
*                                                   ;
a = 100. ; b = 200. ; h = 600. ;t = a / 100.        ;
densite  t ; pf = (a 0.)                            ;
c1 = (c ( pf moin (t 0.)) pf ( pf plus (0. t)))
      c pf ( pf plus (t 0.))                        ;
sf = cout pf c1;
r1 = t ; rr1 = t                                    ;
repeter bhomo 7;
   ri  =  r1 + ( 0.3 * r1 )                         ;
   rri = rr1 + ri                                   ;
   dens ri                                          ;
   ci =( c ( pf moin (rri 0.)) pf ( pf plus (0. rri)))
         c pf ( pf plus (rri 0.))                   ;
   sf = sf et ( cout c1 ci )                        ;
   c1 = ci ; r1 = ri ; rr1 = rri                    ;
fin bhomo                                           ;
dens (a / 3.)                                       ;
p0  = (0. 0.) ; p1 = (b 0.)                         ;
p2  = p0 plus (0. a) ; p3 = p1 plus (0. a)          ;
pi1 = ci poin 1  ; l1  = pi1 d p0                   ;
n = (nbel l1) * -1                                  ;
pi2 = ci poin 4  ; l2  = pi2 d n p2                 ;
pi3 = ci poin 10 ; l3  = pi3 d n p3                 ;
pi4 = ci poin 13 ; l4  = pi4 d   p1                 ;
ci  = inve ci    ;ligh = p2  d   p3                 ;
sc1 = dall l1 (p0 d p2) (inve l2) (ci comp pi2 pi1) ;
sc2 = dall l2 ligh (inve l3) (ci comp pi3 pi2)      ;
sc3 = dall l3 (p3 d p1) (inve l4) (ci comp pi4 pi3) ;
sc  = sc1 et sc2 et sc3                             ;
dens (a / 2.)                                       ;
mrest  = ligh tran (0. (h - a)) dini (a / 2.)       ;
lihaut = mrest cote 3                               ;
phd    = lihaut poin init                           ;
ccp    = sf et sc et mrest                          ;
elim  ccp  0.001                                    ;
cccp   = cont ccp                                   ;
lifis  = cccp comp pf p0                            ;
libas  = cccp comp p1 pf                            ;
licot  = ccp  poin droi (0. 0.) (0. 100.) 0.001     ;
licot  = cccp elem appuy stric licot                ;
SI ( NEG GRAPH 'N' )                                ;
    TRAC  CCP                                       ;
FINSI                                               ;
*                                                   ;
*------------------- CREATION DU MODELE --------------;
*                                                   ;
objaf = modl ccp mecanique elastique isotrope       ;
*                                                   ;
*----------- DEFINITION DES CARACTERISTIQUES ---------;
*-----------   MATERIELLES ET GEOMETRIQUES   ---------;
*                                                   ;
mat = matr objaf  YOUN 2.e4  NU 0.3                 ;
*                                                   ;
*-------- CALCUL DES RIGIDITES ELEMENTAIRES ----------;
*--------     ET DEFINITION DES BLOCAGES    ----------;
*                                                   ;
rig  = rigi objaf mat                               ;
cdl1 = bloq Ux licot                                ;
cdl2 = bloq Uy libas                                ;
*                                                   ;
*-------------- DEFINITION DU CHARGEMENT -------------;
*                                                   ;
F = pres mass objaf -1. lihaut                      ;
*                                                   ;
*-------- RESOLUTION ET CALCUL DES CONTRAINTES -------;
*                                                   ;
U    = reso (rig et cdl1 et cdl2 ) F                ;
SIG  = sigm mat objaf U                             ;
*                                                   ;
*------ INITIALISATION DE LA TABLE EN ENTREE ---------;
*-----       DE LA PROCEDURE G_THETA        ---------;
*                                                   ;
SUPTAB = TABLE ;
SUPTAB.'OBJECTIF' = MOT 'J';
SUPTAB.'LEVRE_SUPERIEURE' = lifis;
SUPTAB.'FRONT_FISSURE' = PF ;
SUPTAB.'MODELE' = objaf;
SUPTAB.'CARACTERISTIQUES' = mat;
SUPTAB.'SOLUTION_RESO' = u;
SUPTAB.'CHARGEMENTS_MECANIQUES' = F;
SUPTAB.'COUCHE' = 4;
*                                                   ;
*--------------- APPEL A LA PROCEDURE G_THETA --------;
*                                                   ;
SAUT PAGE                                           ;
G_THETA suptab                                      ;
*                                                   ;
*----- RECUPERATION DU RESULTAT ET COMPARAISONS ------;
*                                                   ;
G   = suptab.'RESULTATS'                            ;
Kth = 20.8426                                       ;
K = (20000. * G / (1 - (0.3**2)))**0.5              ;
mess '  '                                           ;
mess '  '                                           ;
mess 'taux de restitution energetique  G :' G       ;
mess 'solution calculee            Kcal :' K        ;
mess 'solution analytique          Kthe :' Kth      ;
*                     CODE FONCTIONNEMENT           ;
RESI = abs (( k - Kth ) / Kth )                     ;
SI (RESI <  5E-2)                                   ;
    ERRE  0                                         ;
SINO                                                ;
    ERRE  5                                         ;
FINSI                                               ;
fin;
```

6.4 Test rupt4 Description sheet

Test name rupt4

Calculation type MECHANICS ELASTIC

Finite element type QUA8 PLANE STRAIN MODE

Topic THE USE OF G-INTEGRAL FOR A SIMPLE PLATE WITH A CENTRAL CRACK SUBJECTED TO INTERNAL PRESSURE

The structure is a solid plate containing a central crack . This structure is subjected to a uniform internal pressure.
Due to symmetry only 1/4 of the structure is analysed. The CASTEM solution for the stress intensity factor K is given by the relationship :

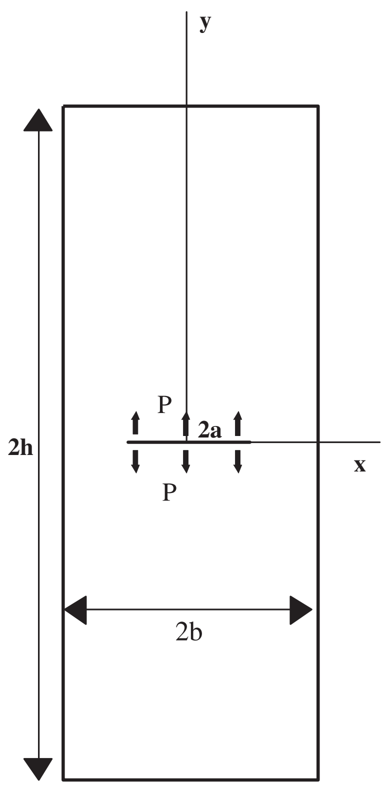
$K = (\sqrt{EG/1-\nu^2})$ *G is the energy release rate*

Finally this CASTEM K value for the stress intensity factors is compared with the result obtained from the analytical formula defined by the relationship :

$$K_0 = \sigma\sqrt{\pi a}$$

Reference CASTEM Test NAFEMS : D.P.Rooke and D. J. Cartwright in Copendium of Intensity Factors HMSO (1976) : rupt4 Modélisation des structures élastiques dans CASTEM 2000.

Version 97' customer version **Model description**



GEOMERTY :
Width of the edge : 2b = 1200 mm
Length of the edge : 2h = 2400 mm
Length of the crack : 2a = 200 mm

MATERIAL CHARACTERISTICS :
E = 20000 daN mmˆ(-2)
nu = 0.3

LOADING CONDITIONS :
1- Uniform internal pressure
P = 1 daN mmˆ(-2)

Solid-Plate with a central crack subject to uniform internal pressure

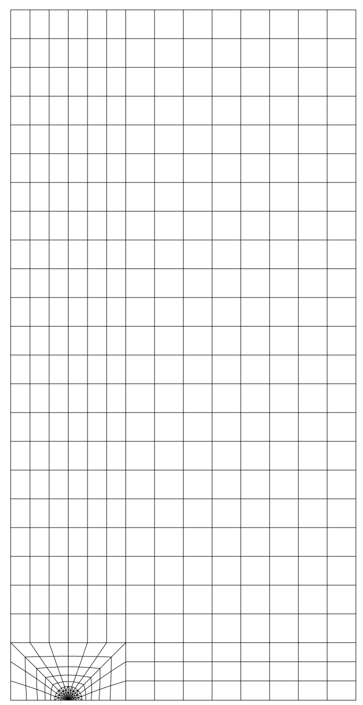
Test rupt4

RESULTS

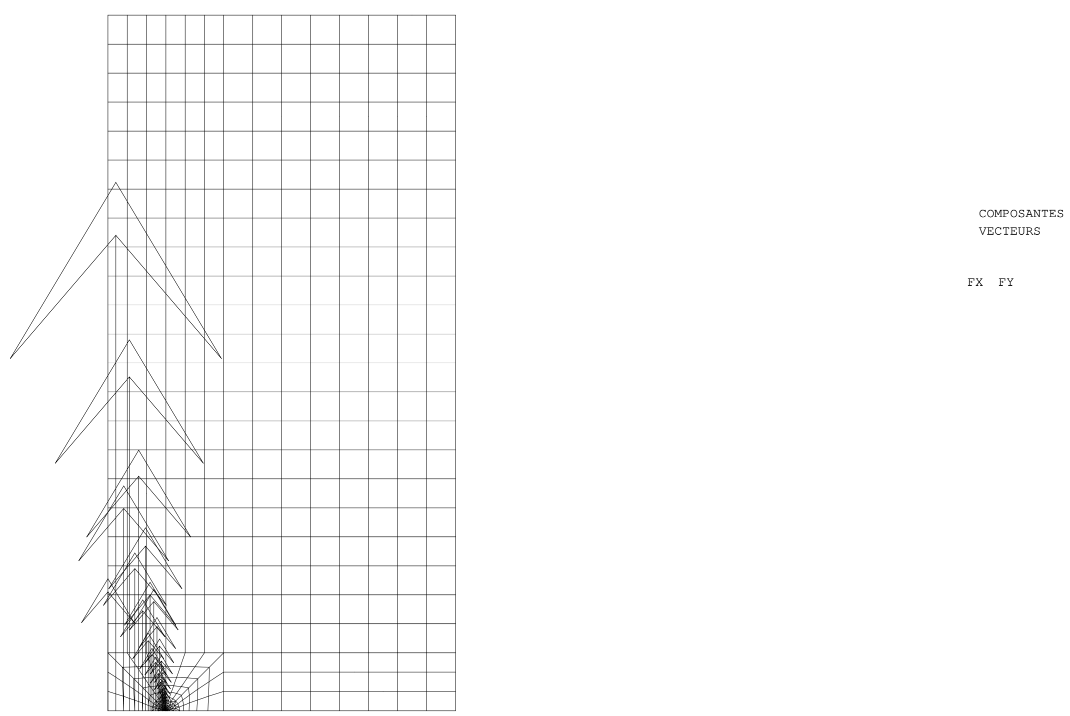
Results

	$K1(daNmm^{-3/2})$
Reference	17.725
Castem	17.616

CASTEM FIGURES



GIBI FECIT



FIELD OF PRESSURE AT THE CRACK TIP

```
*          Test Rupt4.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0                                       ;
GRAPH = 'N'                                         ;
SAUT PAGE                                           ;
*                                                   ;
*****************************************************;
*                                                   ;
*                                                   ;
*             QUALIFICATION DU CALCUL DE G          ;
*             EN ELASTICITE LINEAIRE SUR            ;
*             UNE PLAQUE A FISSURE INTERNE          ;
*             SOUMISE A UNE PRESSION CONSTANTE      ;
*                                                   ;
*                                                   ;
* le calcul est compare a celui obtenu par          ;
* ROOKE et CARTWRIGHT                               ;
* dans Compendium of Stress Intensity Factors       ;
*                                                   ;
*****************************************************;
*                                                   ;
OPTION DIME 2 ELEM QUA8 MODE PLAN DEFO              ;
*                                                   ;
*----------------- DEFINITION DU MAILLAGE ------------;
*                                                   ;
a = 100. ; b = 200. ; h = 1200.                     ;
m = 600 ; t = a / 100                               ;
densite  t ; pf = (a 0.)                            ;
c1 = (c ( pf moin (t 0.)) pf ( pf plus (0. t)))
      c pf ( pf plus (t 0.))                        ;
sf = cout pf c1;
r1 = t ; rr1 = t                                    ;
repeter bhomo 7;
   ri  =  r1 + ( 0.3 * r1 )                         ;
   rri = rr1 + ri                                   ;
   dens ri                                          ;
   ci = ( c ( pf moin (rri 0.)) pf ( pf plus (0. rri)))
         c pf ( pf plus (rri 0.))                   ;
   sf = sf et ( cout c1 ci )                        ;
   c1 = ci ; r1 = ri ; rr1 = rri                    ;
fin bhomo                                           ;
dens (a / 3.)                                       ;
p0  = (0. 0.) ; p1 = (b 0.)                         ;
p2  = p0 plus (0. a) ; p3 = p1 plus (0. a)          ;
pi1 = ci poin 1  ; l1  = pi1 d p0                   ;
n = (nbel l1) * -1                                  ;
pi2 = ci poin 4  ; l2  = pi2 d n p2                 ;
pi3 = ci poin 10 ; l3  = pi3 d n p3                 ;
pi4 = ci poin 13 ; l4  = pi4 d   p1                 ;
ci  = inve ci    ;ligh = p2  d   p3                 ;
sc1 = dall l1 (p0 d p2) (inve l2) (ci comp pi2 pi1) ;
sc2 = dall l2 ligh (inve l3) (ci comp pi3 pi2)      ;
sc3 = dall l3 (p3 d p1) (inve l4) (ci comp pi4 pi3) ;
sc  = sc1 et sc2 et sc3                             ;
dens (a / 2.)                                       ;
ligh1  = p1 d p3                                    ;
mrest1 = ligh1  tran ((m-b) 0.) dini (a/2.)         ;
ala    = mrest1 cote 3                              ;
al     = mrest1 cote 2                              ;
mrest  = ligh tran (0. (h - a)) dini (a / 2.)       ;
mrest2 = al   tran (0. (h - a)) dini (a / 2.)       ;
lihaut1= mrest cote 3                               ;
lihaut = mrest cote 3                               ;
lihaut2= mrest2 cote 3                              ;
lihaut = lihaut1 et lihaut2                         ;
ccp    = sf et sc et mrest et mrest1 et mrest2      ;
elim  ccp  0.001                                    ;
cccp   = cont ccp                                   ;
pj     = ala poin fina                              ;
lifis  = cccp comp pf p0                            ;
libas  = cccp comp pj pf                            ;
licot  = ccp poin droi (0. 0.) (0. 100.) 0.001      ;
licot  = cccp elem appuy stric licot                ;
SI ( NEG GRAPH 'N' )                                ;
     TRAC  CCP                                      ;
FINSI                                               ;
*                                                   ;
*------------------- CREATION DU MODELE -------------;
*                                                   ;
objaf = modl ccp mecanique elastique isotrope       ;
*                                                   ;
*---------- DEFINITION DES CARACTERISTIQUES ----------;
*-------------   MATERIELLES ET GEOMETRIQUES   -------;
*                                                   ;
mat = matr objaf  YOUN 2.e4  NU 0.3                 ;
*                                                   ;
*-------- CALCUL DES RIGIDITES ELEMENTAIRES ----------;
*------     ET DEFINITION DES BLOCAGES    ------------;
*                                                   ;
rig  = rigi objaf mat                               ;
cdl1 = bloq Uy libas                                ;
cdl2 = bloq Ux licot                                ;
*                                                   ;
*------------------ DEFINITION DU CHARGEMENT ---------;
*                                                   ;
Fp = pres mass objaf 1. lifis                       ;
*                                                   ;
*-------- RESOLUTION ET CALCUL DES CONTRAINTES -------;
*                                                   ;
U    = reso (rig et cdl1 et cdl2 ) Fp               ;
SIG  = sigm mat objaf U                             ;
*                                                   ;
*----- INITIALISATION DE LA TABLE EN ENTREE ----------;
*-----       DE LA PROCEDURE G_THETA        ---------;
*                                                   ;
SUPTAB = TABLE ;
SUPTAB.'OBJECTIF' = MOT 'J';
SUPTAB.'LEVRE_SUPERIEURE' = lifis;
SUPTAB.'FRONT_FISSURE' = PF ;
SUPTAB.'MODELE' = objaf;
SUPTAB.'CARACTERISTIQUES' = mat;
SUPTAB.'SOLUTION_RESO' = u;
SUPTAB.'CHARGEMENTS_MECANIQUES' = Fp;
SUPTAB.'COUCHE' = 4;
*                                                   ;
*---------- APPEL A LA PROCEDURE G_THETA -------------;
*                                                   ;
SAUT PAGE                                           ;
G_THETA suptab                                      ;
*                                                   ;
*----- RECUPERATION DU RESULTAT ET COMPARAISONS ------;
*                                                   ;
G   = suptab.'RESULTATS'                            ;
Kth = 17.7245                                       ;
K = (20000. * G / (1 - (0.3**2)))**0.5              ;
mess '  '                                           ;
mess '  '                                           ;
mess 'taux de restitution energetique  G :' G       ;
mess 'solution calculee            Kcal :' K        ;
mess 'solution analytique          Kthe :' Kth      ;
*                    CODE FONCTIONNEMENT            ;
RESI = abs (( k - Kth ) / Kth )                     ;
SI (RESI <  5E-2)                                   ;
    ERRE  0                                         ;
SINO                                                ;
```

```
    ERRE  5                                            ;
FINSI                                                  ;
fin;
```

6.5 Test rupt5 Description sheet

Test name rupt5

Calculation type MECHANICS ELASTIC

Finite element type QUA8 PLANE STRAIN MODE

Topic THE USE OF J-INTEGRAL FOR A SIMPLE AXISYMETRICAL TUBE WITH A CENTRAL CRACK STRIP SUBJECTED TO UNIFORM INTERNAL PRESSURE

The structure is a solid cylinder tub containing a central crack . This structure is subject to an internal pressure. Due to symmetry only 1/4 of the structure is analysed. The CASTEM solution for the stress intensity factors K is given by the relationship (axisymetrical model) :

$K = (\sqrt{EG/1-\nu^2})$ $\qquad$ G *is the energy release rate*

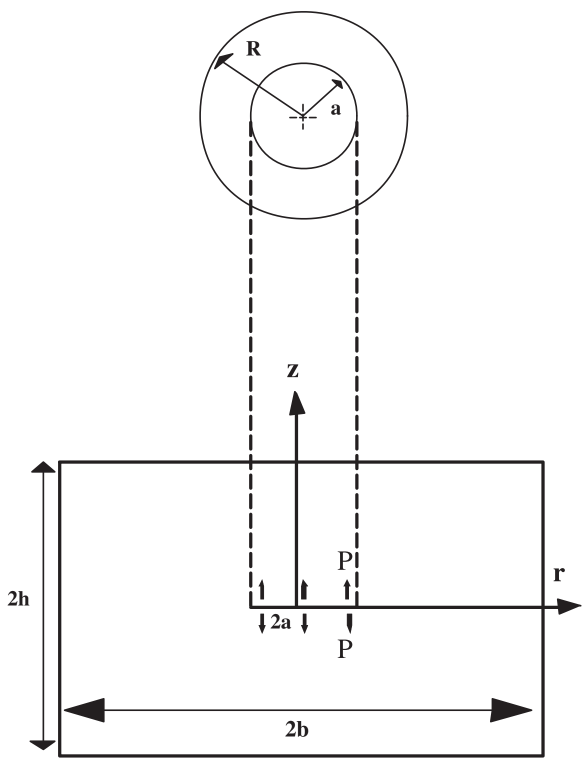
Finally this CASTEM K value for the stres intensity factor is compared with the result obtained from the analytical formula defined by the relationship :

$$\frac{2}{\sqrt{\pi a}} \int_0^a \frac{r p(r)}{\sqrt{a^2 - r^2}} dr$$

$$P(r) = 1 - \frac{r}{a}$$

Reference CASTEM Test NAFEMS : D.P.Rooke and D. J. Cartwright in Copendium of Intensity Factors HMSO (1976) : rupt5 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97'

customer version **Model description**



Axisymetrical tub with a central crack
subject to internal pressure

GEOMERTY :

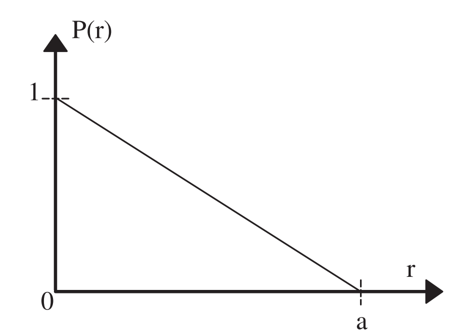
Width of the edge : 2b = 2000 mm
Length of the edge : 2h = 1000 mm
Length of the crack : 2a = 200 mm

MATERIAL CHARACTERISTICS :

E = 20000 daN mmˆ(-2)
nu = 0.3

LOADING CONDITIONS :

1- Internal pressure
P(r) = 1 - (r/a)



Test rupt5

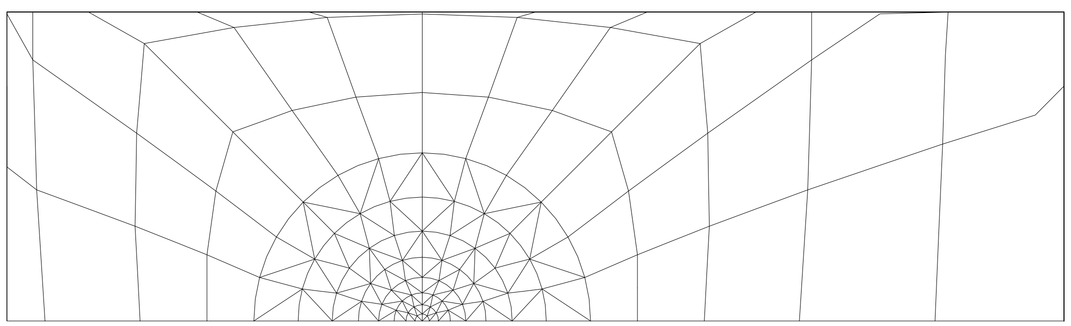
Results

RESULTS

	$K1(daNmm^{-3/2})$
Reference	2.4215
Castem	2.4098

CASTEM FIGURES

MESH OF THE CYLINDER TUB



MESH OF THE CYLINDER TUB

```
*          Test Rupt5.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0                                       ;
GRAPH = 'N'                                         ;
SAUT PAGE                                           ;
*                                                   ;
*****************************************************;
*                                                   ;
*                                                   ;
*             QUALIFICATION DU CALCUL DE G          ;
*             EN ELASTICITE LINEAIRE SUR            ;
*             UN TUBE A FISSURE INTERNE             ;
*             SOUMISE A UNE PRESSION LINEAIRE       ;
*                                                   ;
*                                                   ;
* le calcul est compare a celui obtenu par          ;
* ROOKE et CARTWRIGHT                               ;
* dans Compendium of Stress Intensity Factors       ;
*                                                   ;
*****************************************************;
*                                                   ;
OPTION DIME 2 ELEM QUA8 MODE AXIS                   ;
*                                                   ;
*------------- DEFINITION DU MAILLAGE ---------------;
*                                                   ;
a = 100. ; b = 200. ; h = 500.                      ;
m = 1000 ; t = a / 100                              ;
densite  t ; pf = (a 0.)                            ;
c1 = (c ( pf moin (t 0.)) pf ( pf plus (0. t)))
      c pf ( pf plus (t 0.))                        ;
sf = cout pf c1;
r1 = t ; rr1 = t                                    ;
repeter bhomo 7;
   ri  =  r1 + ( 0.3 * r1 )                         ;
   rri = rr1 + ri                                   ;
   dens ri                                          ;
   ci = ( c ( pf moin (rri 0.)) pf ( pf plus (0. rri)))
         c pf ( pf plus (rri 0.))                   ;
   sf = sf et ( cout c1 ci )                        ;
   c1 = ci ; r1 = ri ; rr1 = rri                    ;
fin bhomo                                           ;
dens (a / 3.)                                       ;
p0  = (0. 0.) ; p1 = (b 0.)                         ;
p2  = p0 plus (0. a) ; p3 = p1 plus (0. a)          ;
pi1 = ci poin 1  ; l1  = pi1 d p0                   ;
n = (nbel l1) * -1                                  ;
pi2 = ci poin 4  ; l2  = pi2 d n p2                 ;
pi3 = ci poin 10 ; l3  = pi3 d n p3                 ;
pi4 = ci poin 13 ; l4  = pi4 d   p1                 ;
ci  = inve ci    ;ligh = p2  d   p3                 ;
sc1 = dall l1 (p0 d p2) (inve l2) (ci comp pi2 pi1) ;
sc2 = dall l2 ligh (inve l3) (ci comp pi3 pi2)      ;
sc3 = dall l3 (p3 d p1) (inve l4) (ci comp pi4 pi3) ;
sc  = sc1 et sc2 et sc3                             ;
dens (a / 2.)                                       ;
ligh1  = p1 d p3                                    ;
mrest1 = ligh1  tran ((m-b) 0.) dini (a/2.)         ;
ala    = mrest1 cote 3                              ;
al     = mrest1 cote 2                              ;
mrest  = ligh tran (0. (h - a)) dini (a / 2.)       ;
mrest2 = al   tran (0. (h - a)) dini (a / 2.)       ;
lihaut1= mrest cote 3                               ;
lihaut = mrest cote 3                               ;
lihaut2= mrest2 cote 3                              ;
lihaut = lihaut1 et lihaut2                         ;
ccp    = sf et sc et mrest et mrest1 et mrest2      ;
elim  ccp  0.001                                    ;
```

```
cccp   = cont ccp                                   ;
pj     = ala poin fina                              ;
lifis  = cccp comp pf p0                            ;
libas  = cccp comp pj pf                            ;
SI ( NEG GRAPH 'N' )                                ;
     TRAC  CCP                                      ;
FINSI                                               ;
*                                                   ;
*-------------- CREATION DU MODELE ------------------;
*                                                   ;
objaf = modl ccp mecanique elastique isotrope       ;
*                                                   ;
*--------- DEFINITION DES CARACTERISTIQUES -----------;
*-------------   MATERIELLES ET GEOMETRIQUES   -------;
*                                                   ;
mat = matr objaf  YOUN 2.e4  NU 0.3                 ;
*                                                   ;
*--------- CALCUL DES RIGIDITES ELEMENTAIRES ---------;
*------ -     ET DEFINITION DES BLOCAGES    ----------;
*                                                   ;
rig  = rigi objaf mat                               ;
cdl1 = bloq Uz libas                                ;
cdl2 = bloq Uz lihaut                               ;
*                                                   ;
*------------- DEFINITION DU CHARGEMENT -------------;
*                                                   ;
pchx = coor 1 lifis                                 ;
pcha = manu chpo lifis 1 scal a                     ;
pchx = nomc 'P' (pcha - pchx)                       ;
pcht = pchx * (1/a)                                 ;
pch0 = pcht *0.                                     ;
Fp   = pres mass objaf pcht                         ;
*                                                   ;
*------- RESOLUTION ET CALCUL DES CONTRAINTES --------;
*                                                   ;
U    = reso (rig et cdl1 et cdl2 ) Fp               ;
SIG  = sigm mat objaf U                             ;
*                                                   ;
*----- INITIALISATION DE LA TABLE EN ENTREE ----------;
*------       DE LA PROCEDURE G_THETA        --------;
*                                                   ;
SUPTAB = TABLE ;
SUPTAB.'OBJECTIF' = MOT 'J';
SUPTAB.'LEVRE_SUPERIEURE' = lifis;
SUPTAB.'FRONT_FISSURE' = PF ;
SUPTAB.'MODELE' = objaf;
SUPTAB.'CARACTERISTIQUES' = mat;
SUPTAB.'SOLUTION_RESO' = u;
SUPTAB.'CHARGEMENTS_MECANIQUES' = Fp;
SUPTAB.'COUCHE' = 4;
*                                                   ;
*----------- APPEL A LA PROCEDURE G_THETA ------------;
*                                                   ;
SAUT PAGE                                           ;
G_THETA suptab                                      ;
*                                                   ;
*------ RECUPERATION DU RESULTAT ET COMPARAISONS -----;
*                                                   ;
G   = suptab.'RESULTATS'                            ;
Kth = 2.4215                                        ;
K = (20000. * G / (1 - (0.3**2)))**0.5              ;
mess '  '                                           ;
mess '  '                                           ;
mess 'taux de restitution energetique  G :' G       ;
mess 'solution calculee            Kcal :' K        ;
mess 'solution analytique          Kthe :' Kth      ;
*                  CODE FONCTIONNEMENT              ;
RESI = abs (( k - Kth ) / Kth )                     ;
```

```
SI (RESI <  5E-2)                                     ;
    ERRE  0                                           ;
SINO                                                  ;
    ERRE  5                                           ;
FINSI                                                 ;
fin;
```

6.6 Test rupt6 Description sheet

Test name rupt6

Calculation type MECHANICS ELASTIC

Finite element type CU20 MODE TRID

Topic STRESS INTENSITY FACTOR FOR A CIRCULAR PLANE CRACK IN AN INFINITE SOLID CYLINDER SUBJECTED TO TENSILE STRESS

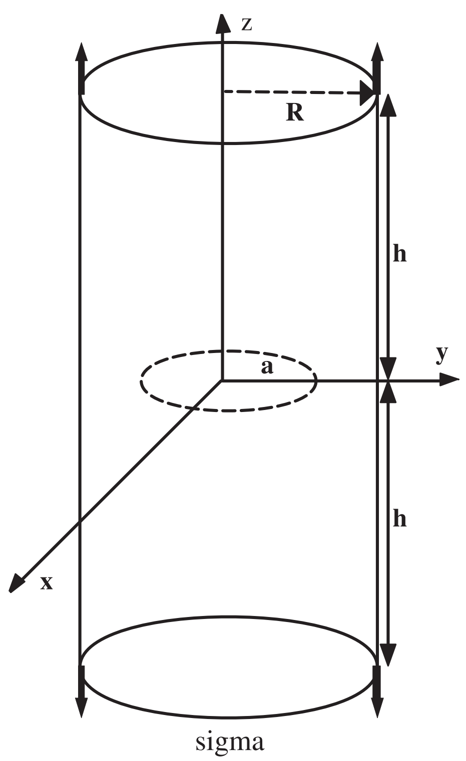
The structure is an infinite solid cylinder tube enclosing a central circular crack . This structure is subject to tensile stress. Due to symmetry only 1/15 of half of the cylinder is analysed. The CASTEM solution for the stress intensity factors K is given with the displacement method by the relationship (threedimensional analysis) :

$$K = \frac{E'}{4}(\sqrt{\frac{2\Pi}{r-\beta}})v \qquad E' = \sqrt{E/1-\nu^2} \quad (planestrain) \quad \beta = -\frac{d}{c}$$

c and d are the coefficients of the line : $v^2 = cr + d$
Finally this CASTEM K value for the stress intensity factor is compared with the result obtained with the analytical formula defined by the relationship : $K_0 = 2\sigma\sqrt{a}\Pi$

Reference CASTEM Test NAFEMS : D.P.Rooke and D. J. Cartwright in Copendium of Intensity Factors HMSO (1976) : rupt6 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97'

customer version **Model description**



GEOMERTY :
Radius of the Cylinder : R = 100 mm
Height of the Cylinder : 2h = 200 mm
Radius of the Crack : a = 18 mm

MATERIAL CHARACTERISTICS :
E = 20000 daN mm^(-2)
nu = 0.3

LOADING CONDITIONS :
1- Uniform load (Tensile stress)
Sigma = 200 daN mm^(-2)

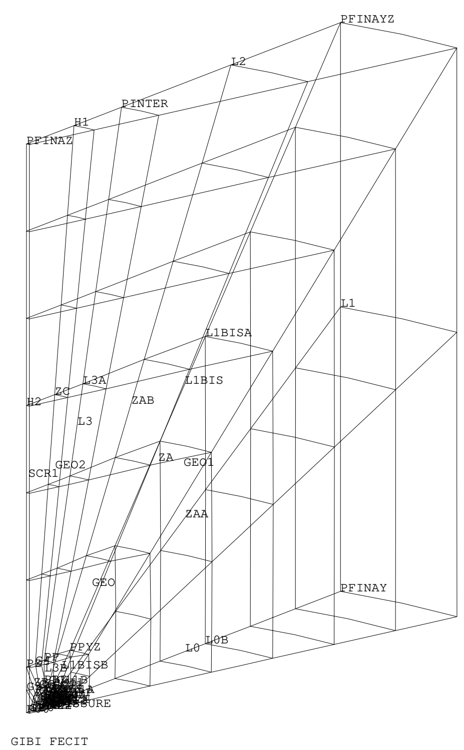
Cylinder with circumference crack
subject to Tensile Stress test

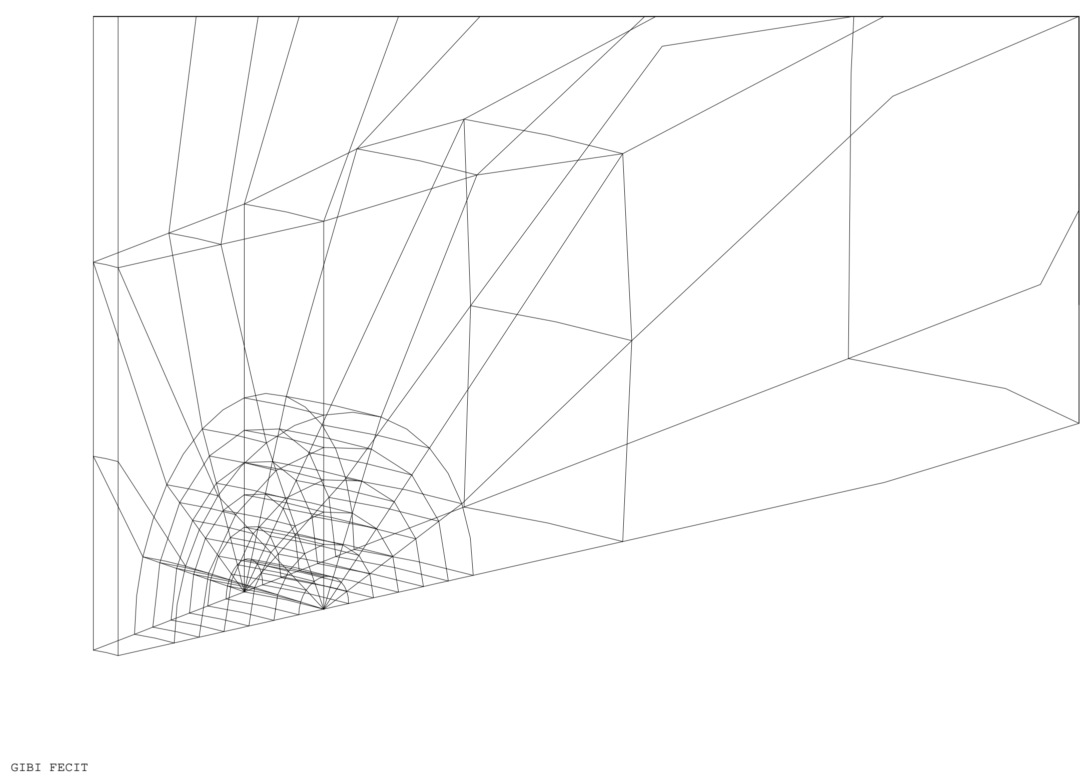
Test rupt6 **Results**

RESULTS

	$K1(topnode)(daNmm^{-3/2})$	$K1(middelenode)(daNmm^{-3/2})$
Reference	607.41	580.48
Castem	638.47	638.47

CASTEM FIGURES





```
*           Test Rupt6.dgibi: Jeux de données
*           --------------------------------
*
******************************************************
***  CALCUL DU FACTEUR D'INTENSITE DE CONTRAINTES  ****
***  PAR LA METHODE DES DEPLACEMENTS ET PAR LA     ****
***  METHODE G_THETA POUR UNE FISSURE CIRCULAIRE   ****
***  PLANE DANS UN MILIEU INFINI CHARGE EN         ****
***  TRACTION UNIFORME                             ****
***  HAUTEUR DU CYLINDRE : 200 mm  ;               ****
***  RAYON : 100 mm                                ****
***  RAYON DE FISSURE    :   8 mm  ;               ****
***  CONTRAINTE UNIFORME : 200 daN / mm2           ****
***  MODELISATION  : TRANCHE DE 15 DE LA 1/2      ****
***                  HAUTEUR DU CYLINDRE           ****
******************************************************
option echo 0 dime 3 elem cu20 mode trid       ;

**----------------- MAILLAGE  ------------------;
  dens 0.5                                     ;
  oeilz = -500   500   -500                    ;
  oeilx = -500   00   00                       ;
  a0    = 8                                    ;
  c0    = 16.                                  ;
  b0    = 100.                                 ;
  p0    =    0     0     0                     ;
  pa    =   a0     0     0                     ;
  pb    =    0    a0     0                     ;
  pa1   =(1.5*a0)  0     0                     ;
  pa0   =(0.5*a0)  0     0                     ;
  pb0   =    0    (0.5*a0)   0                 ;
  pb1   =    0    (1.5*a0) 0                   ;

* nbrz nombre d element autour du front de fissu;
  nbrz  =   6                                  ;

*MAILLAGE DU FRONT DE FISURE                    ;

pbz= (0.  a0  (0.5*a0))                        ;
pbz1=(0. (a0 *(1.- (0.5 / nbrz))) 0.)          ;
pbz2=(0.  a0  (0.5*a0/ nbrz))                  ;
pbz3=(0. (a0 *(1.+ (0.5 / nbrz))) 0.)          ;
aa  = a0*0.5 * (2**-0.5)                       ;
pmi1= 0. (a0 + aa) aa                          ;
pmi2= 0. (a0 - aa) aa                          ;

 cc11a  = c (2) pb1  pb pmi1                   ;
 cc11b  = c (2) pmi1 pb pbz                    ;
 cc11   = cc11a et cc11b                       ;


 cc12a  = c (2) pbz  pb pmi2                   ;
 cc12b  = c (2) pmi2 pb pb0                    ;
 cc12   = cc12a et cc12b                       ;

 cc1   = cc11 et cc12                          ;

 cc21  = c (4) pbz1 pb pbz2                    ;
 cc22  = c (4) pbz2 pb pbz3                    ;
 cc2   = cc21 et cc22                          ;

 ligz = d (nbrz - 1) pb0 pbz1                  ;
 ligz1= d (nbrz - 1)  pbz3 pb1                 ;
 ligz2= d (nbrz - 1) pbz pbz2                  ;
 scz1 = dall cc11 ligz2 cc22 ligz1             ;
 scz2 = dall cc12 ligz  cc21 (inve ligz2)      ;

 scz1=scz1 et (cout cc22 pb)                   ;
 scz2=scz2 et (cout cc21 pb)                   ;
 scz =scz1 et scz2                             ;
 elim (scz1 et scz2)  0.001                    ;

* SURFACE YZ                                   ;
pfinay = 0. b0 0.                              ;
pfinaz = 0.  2.5 b0                            ;
pfinayz= 0. b0 b0                              ;
pinter = 0. (2*c0) b0                          ;
pp = 0. a0 a0                                  ;
pz = 0. 2.5 a0                                 ;
p00= 0. 2.5 0.                                 ;
ppp= 0. (2*a0) 0.                              ;
ppyz= 0. (2*a0) a0                             ;

l0a= d (1) pb1 ppp                             ;
l0b= d (6) ppp pfinay                 ;
l0 = l0a et l0b                                ;
l1 = d (2) pfinay  pfinayz                     ;
l1bisa = d (6) pfinayz ppyz                    ;
l1bisb = d (1)    ppyz    pmi1                 ;
l1bis  = l1bisa et l1bisb                      ;
l2 = d (2) pfinayz  pinter                     ;
l3a = d (6) pinter pp                 ;
l3b = d (1) pp pbz                             ;
l3 = l3a et l3b                                ;
zaa = dall (inve cc11a) l0 l1 l1bis            ;
zab = dall l1bis cc11b (inve l3) (inve l2)     ;
za  = zaa et zab                               ;

g2= d (1) pb0 p00                              ;
g3= d (2) p00 pz                               ;
g4= d (2) pz pp                                ;
zb =dall (g4 et g3) g2  cc12  l3b              ;

h1 = d (2) pinter pfinaz                       ;
h2 = d (6) pfinaz pz                           ;
zc = dall h2  g4  l3a  h1                      ;

scr1 = h2 et (inve g3)                         ;

*CREATION DU VOLUME                            ;
scr1 = scr1 rota 1 (15) (0. 0 -500)
       (0 0 500.) coul roug                    ;

geo1 = za volu 1 'ROTA' (15)
          (0. 0. -500) (0. 0. 500.)            ;

geo2 = (zb et zc) volu 1 'ROTA' (15)
          (0. 0. -500) (0. 0. 500.)            ;

geo3 = scz1 volu 1 'ROTA' (15)
          (0. 0. -500) (0. 0. 500.)            ;
geo4 = scz2 volu 1 'ROTA' (15)
          (0. 0. -500) (0. 0. 500.)            ;
i = face 3 geo3                                ;
i = i et (face 3 geo4)                         ;

pppp = i poin cylin (0 0 -100) (0 0 100) pb    ;
aa0= elem i appu larg pppp                     ;
aa1= poin aa0 plan p0 (0. 100 0.) (100 0 0)    ;
cfis = elem aa0 appu stric aa1                 ;
a1 = enve geo3                                 ;
a2 = enve geo4                                 ;
aa1= poin a1 plan p0 (0. 100 0.) (100 0 0)     ;
aa2= poin a2 plan p0 (0. 100 0.) (100 0 0)     ;
aa11 = elem a1 appu stric aa1                  ;
aa21 = elem a2 appu stric aa2                  ;
```

```
aa12 = cont aa11                                 ;
aa22 = cont aa21                                 ;
elim (aa12 et aa22) 0.001                        ;
cfissure = elem aa12 appu stric aa22;

geo = geo1 et geo2 et geo3 et geo4               ;

elim (geo et scr1 et cfissure et cfis) 0.001    ;

*FINITION DU MAILLAGE                            ;
*pres de l axe z                                 ;
 c1 = cote (2) scr1                              ;
 pfi  = c1 poin final                            ;
 c1x  = droi (1) pfi p0                          ;
 b1 = droi (1) p00 p0                            ;

 su0 = surf ( c1x et c1  et b1) plane            ;
 su1 = su0 volu (2) tran (0. 0. a0)              ;
 ss = su0 plus (0. 0. b0)                        ;
 su2= ss volu (6) tran (0. 0. (a0 - b0))         ;
 su = su1 et su2  coul vert                      ;

*le cylindre en entier                           ;
cub = geo et su                                  ;
elim cub 0.001                                   ;
cub = rege cub                                   ;
tot = enve cub                                   ;

*surface yz                                      ;
poyz = poin tot plan p0 (0. 100 100) (0. 0. 100);
suryz = elem tot appu stric poyz                 ;

*surface z=100                                   ;
poz100= poin tot plan (0. 0. 100)
        (100 0. 100) (0. 100 100)                ;
surzz = elem tot appu stric poz100               ;

*surfac xy                                       ;
poxy = poin tot plan p0 (0. 100 0.) (100 0 0)   ;
surxy= elem tot appu stric poxy                  ;
az = geo2 et geo4 et su                          ;
az = enve az                                     ;
aze= az poin plan p0  (0. 100 0.) (100 0 0)      ;
aze=elem az appu stric aze                       ;
co = cont aze;
aze = surxy incl aze;
facxy = surxy diff aze;

*--------------CONDITION DE SYMETRIE------------;
p = ((-1 * b0 * (sin 15)) (b0 * (cos 15)) 0.);
condi1 =symt depl p0 (0. 0. b0) p cub 0.01;
condi2 =symt depl p0 pfinay pfinayz        suryz;
condi3 =symt depl p0 (b0 b0 0.) (b0 0. 0.) facxy;
condit = condi1 et condi2 et condi3            ;

*----------- CREATION DU MODELE ----------------;

affe1  = modl  geo1 mecanique elastique isotrope;
affe2  = modl  geo2 mecanique elastique isotrope;
affe3  = modl  geo3 mecanique elastique isotrope;
affe4  = modl  geo4 mecanique elastique isotrope;
affe5  = modl  su   mecanique elastique isotrope;

affetot =  affe1 et affe2 et
           affe3 et affe4 et affe5             ;

*---------DEFINITION DU MATERIAU ---------------;
```

```
mate1   =  matr  affe1
           young    20000.
           nu          0.3                      ;
mate2   =  matr  affe2
           young    20000.
           nu          0.3                      ;
mate3   =  matr  affe3
           young    20000.
           nu          0.3                      ;
mate4   =  matr  affe4
           young    20000.
           nu          0.3                      ;
mate5   =  matr  affe5
           young    20000.
           nu          0.3                      ;
matot   =  mate1 et mate2 et
           mate3 et mate4 et mate5              ;

*----------------RIGIDITE-----------------------;

rig1    =  (rigidite matot affetot) et condit   ;
rig2    =  bloque   uz  surzz                   ;
fo1     =  depi rig2 1.                         ;

*------------RESOLUTION ET CONTRAINTES----------;
dep     = resou (rig1 et rig2) fo1              ;
sig     = sigma  matot  affetot dep             ;


*----CALCUL DU FACTEUR D'INTENSITE DE CONTRAINTES----*
*----initialisations des paramètres de procedure-----*
SUPTAB = TABLE;
SUPTAB.'MAILLAGE' = cub;
SUPTAB.'PSF1' = P0;
SUPTAB.'FRTFISS' = elem aa12 appu stri aa22;

*-----APPEL DE LA PROCEDURE SIF ----------------------*
SIF SUPTAB MATOT dep;

*-----APPEL DE LA PROCEDURE G_THETA ------------------*

TABG1 = TABLE;
TABG1.'OBJECTIF' = MOT 'J';
TABG1.'COUCHE' = 3;
TABG1.'FRONT_FISSURE' = cfissure;
TABG1.'LEVRE_SUPERIEURE' = aa21 ;
TABG1.'MODELE' = affetot;
TABG1.'SOLUTION_RESO' = dep;
TABG1.'CHARGEMENTS_MECANIQUES' = fo1;
TABG1.'CARACTERISTIQUES' = matot;
G_THETA TABG1;
SAUT 1 LIGNE;
*****************************************************
********    DEPOUILLEMENT PROCEDURE SIF   *************
*****************************************************
T = INDEX (SUPTAB.K1);
MESS 'K1 NOEUD 1 DU FRONT DE FISSURE (NOEUD SOMMET) : '
       SUPTAB.K1.(T.1);
MESS 'K1 NOEUD 2 DU FRONT DE FISSURE (NOEUD MILIEU) : '
      SUPTAB.K1.(T.2);
*---------TEST D'ERREUR-----------------------------*
EC1 = ((SUPTAB.K1.(T.1)) - 638.47) /
      (SUPTAB.K1.(T.1)) * 100;
EC2 = ((SUPTAB.K1.(T.2)) - 638.47) /
      (SUPTAB.K1.(T.2)) * 100;

SI ((EC1 < 5.5) ET (EC2 < 11)) ;
  MESS 'PROCEDURE <SIF> ERR  0';
```

```
  ERRE 0;
SINON;
  MESS 'PROCEDURE <SIF> ERR  5';
  ERRE 5;
FINSI;
*******************************************************
********    DEPOUILLEMENT PROCEDURE G_THETA   *********
*******************************************************
TBG = TABG1.'RESULTATS';
IND1 = INDE TBG;
GSOM = TBG.(IND1.1);
GMIL = TBG.(IND1.2);
EPRI = 20000. / 0.91;
KSOM = (GSOM * EPRI) ** 0.5;
KMIL = (GMIL * EPRI) ** 0.5;
SAUT 1 LIGNE;
MESS 'K1 NOEUD 1 DU FRONT DE FISSURE (NOEUD SOMMET) : '
KSOM;
MESS 'K1 NOEUD 2 DU FRONT DE FISSURE (NOEUD MILIEU) : '
KMIL;
*---------TEST D'ERREUR-----------------------------*
EC1 = ABS (((KSOM - 638.47) / KSOM) * 100);
EC2 = ABS (((KMIL - 638.47) / KMIL) * 100);

SI ((EC1 < 0.9) ET (EC2 < 0.8)) ;
  MESS 'PROCEDURE <G_THETA> ERR  0';
  ERRE  0;
SINON;
  MESS 'PROCEDURE <G_THETA> ERR  5';
  ERRE  5;
FINSI;
FIN;
```

Test rupt6 **Comments**

DESCRIPTION OF THE SIF PROCEDURE

```
SIF SUPTAB MATOT DEP;
```

- This procedure calculates the stress intensity factor in mode I (possibly in mode II) from the displacements of the sides of the crack. The calculated value is an average of the three points which are closest to the crack tip. This procedure works in two and three dimensions. IN 2D, the problem relative to loading in mixed modes can be solved. In 3D massive elements (for example: 3D massiv elements CUB20), for each node of the crack front, the stress intensity factor is calculated from the displacements of the points located in the plane normal to the crack front at the considered node. (while generating the mesh, make sure that you will have these planes normal to the crack front). If the crack is opening,the plane stress formula is applied to the points located at the surface. In 3D with thin shells, the membrane and bending terms of the stress intensity factor can be computed. The bending term is deduced from the knot rotations inducing a strain at the external boundary of the shells. Mixed mode loadings are allowed.
  Input data :
  SUPTAB : TABLE type object, indexed by words, used to define the calculation options and parameters :
  MATOT : field of material properties
  DEP : displacement field
  SUPTAB : TABLE type object, indexed by words, used to define the calculation options and parameters :
- for this three-dimensional problem with massive elements the parameters of SUPTAB are

```
SUPTAB = TABLE;
SUPTAB.'MAILLAGE' = cub;
SUPTAB.'PSF1' = P0;
SUPTAB.'FRTFISS' = elem aa12 appu stri aa22;
```

'PSF1'=P0; : point of the crack surface that does not pertain to the front
'FRTFISS'= elem aa12 appu stri aa22; : line describing the front crack

- output data
  GOING THROUGH THE SIF-PROCEDURE

```
T = INDEX (SUPTAB.K1);
MESS 'K1 NODE 1 FOR THE CRACK TIP (TOP NODE) : '
      SUPTAB.K1.(T.1);
MESS 'K1 NODE 2 FOR THE CRACK TIP (MIDDEL NODE) : '
      SUPTAB.K1.(T.2);
```

- As output, SUTAB makes it possible to recover the values of the stress intensity factor
  K1 : TABLE containing the values of K1 at each node of the front (here at 2 nodes T.1 and T.2)

6.7 Test rupt7 Description sheet

Test name rupt7

Calculation type MECHANICS ELASTIC

Finite element type QUA8 DIME 2

Topic STRESS INTENSITY FACTOR FOR AN OBLIQUE EDGE CRACK IN A PLATE SUBJECTED TO UNIFORM TENSILE STRESS

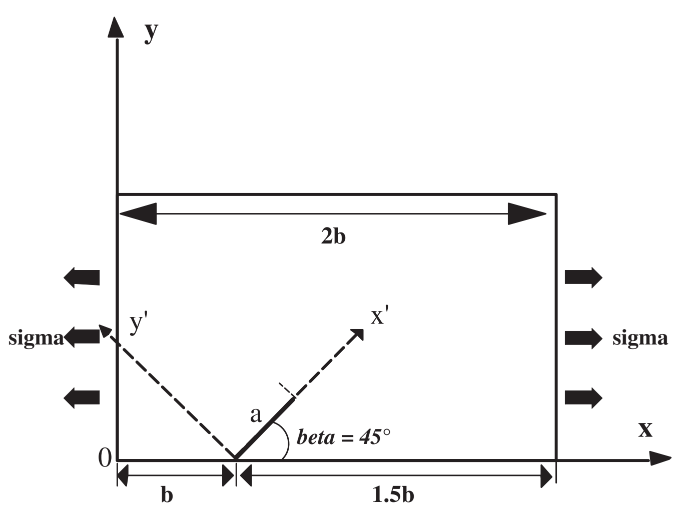
The structure is a finite plate enclosing an oblique edge crack . This structure is subject to uniform tensile stress. The CASTEM solution for the stress intensity factors K is given with the displacements method in 3 nodes at the crack tip (including the mid-node) from the local displacements wich are computed in the local basis $(X'Y')$ for the crack by the relationship :

$$K_I = \frac{E'}{4}(\sqrt{\frac{2\Pi}{r-\beta}})v \qquad K_{II} = \frac{E'}{4}(\sqrt{\frac{2\Pi}{r-\beta'}})u \qquad where \quad E' = \sqrt{E/1-\nu^2} \quad (planestrain)$$

$\beta = -\frac{d}{c}$ c and d are the coefficients of the line : $v^2 = cr + d$
$\beta' = -\frac{b}{a}$ a and b are the coefficients of the line : $u^2 = ar + b$
u : is the displacement following the ox axis.
v : is the displacement following the oy axis .
r : is the distance between the point, where K is computed, and the local origin.
This CASTEM K value for the stress intensity factor is compared to the result obtained with the analytical method .

Reference CASTEM Test NAFEMS : D.P.Rooke and D. J. Cartwright in Copendium of Intensity Factors HMSO (1976) : rupt7 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97'

customer version **Model description**



Tension for a plate with the 45°angled kinked crack

GEOMETRY :
Width of the plate : b = 100 mm
Length of the plate : 2.5b = 250 mm
Length of the crack : a = 50 mm
beta = 45°

MATERIAL CHARACTERISTICS :
E = 20000 daN mm^(-2)
nu = 0.3

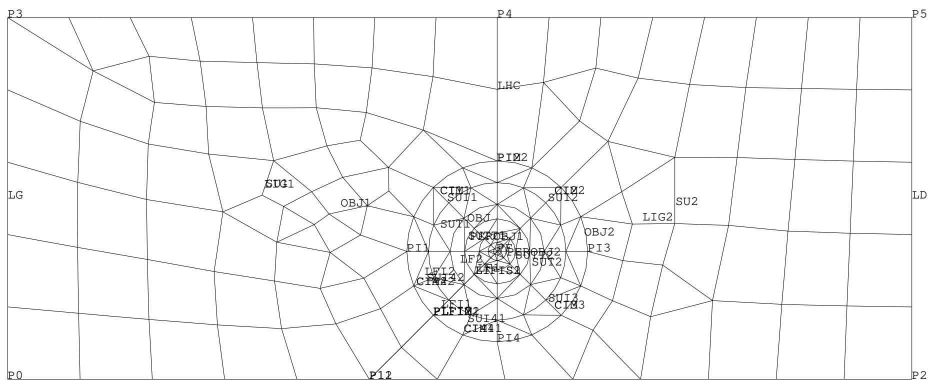
LOADING CONDITIONS :
Tensile Stress :
Sigma = 10 daN mm^(-2)

Test rupt7 **Results**

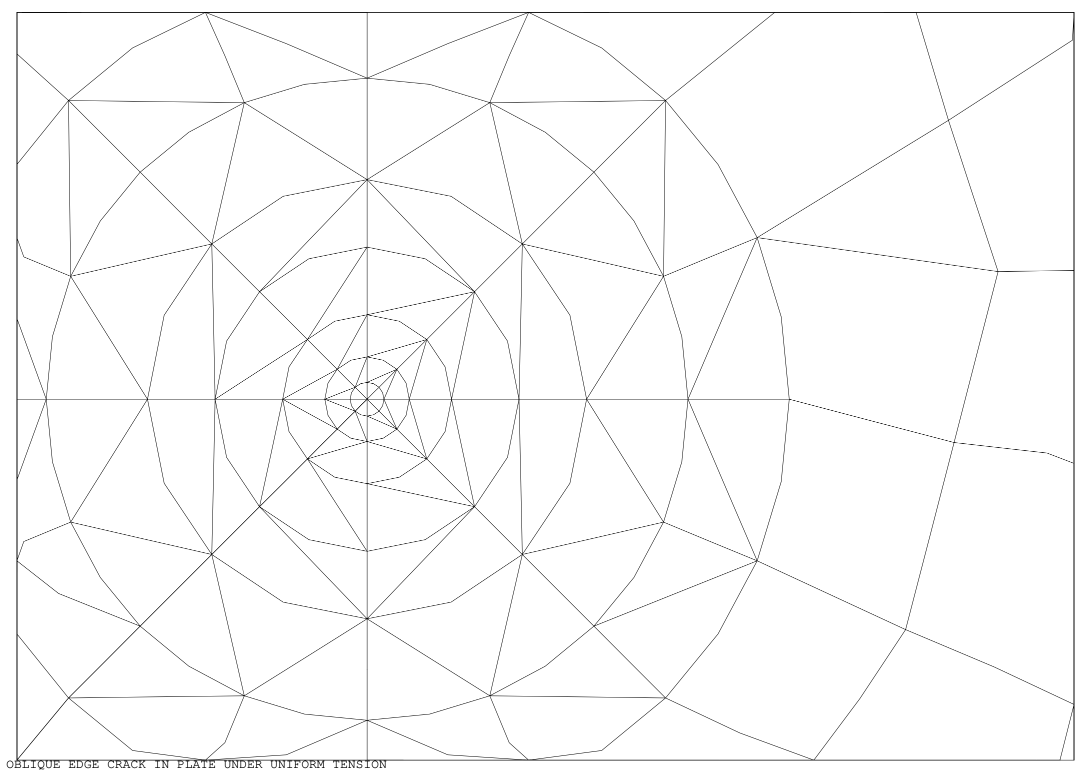
RESULTS

	$K_I(daNmm^{-3/2})$	$K_II(daNmm^{-3/2})$
Reference	150.37	-71.4
Castem	151.2	-71.56

CASTEM FIGURES



OBLIQUE EDGE CRACK IN PLATE UNDER UNIFORM TENSION



OBLIQUE EDGE CRACK IN PLATE UNDER UNIFORM TENSION

```
*          Test Rupt7.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
*****************************************************
*****   CALCUL DU FACTEUR D'INTENSITE DE        *****
*****   CONTRAINTES D'UNE PLAQUE AVEC FISSURE   *****
*****   RECTILIGNE INCLINEE DEBOUCHANTE CHARGEE *****
*****   EN TRACTION UNIFORME                    *****
*****   LARGEUR DE LA PLAQUE : 100 mm ;         *****
*****   LONGUEUR   : 250 mm                     *****
*****   LONGUEUR DE FISSURE : 50 mm   ;         *****
*****   ANGLE / OX : 45                        *****
*****   CONTRAINTE UNIFORME : 10 daN/mm2        *****
*****************************************************
opti dime 2 elem qua8 mode plan defo echo 0;
*--------------DEFINITION DU MAILLAGE-----------------;

p0=0. 0.;p11=100. 0.;p12=p11 plus (0. 0.);p2=250. 0.;
p3=0. 100.;
a=50. * (cos 45);
pf=p11 plus (a a);
p4=(p11 plus (a 0.)) plus (0. 100.);p5=250. 100.;
i=0; pd=prog 1. 2.5 5. 9. 13. 19. 25. ;
cim1=pf;cim2=pf;cim3=pf;cim41=pf;cim42=pf;
plfim1=pf;plfim2=pf;pim2=pf;
repeter bbbb 7;
   i= i + 1;v=extr pd i;
   pi1=pf moin (v 0.);pi2=pf plus (0. v);
   pi3=pf plus (v 0.);pi4=pf moin (0. v);
   ci4=cerc  2 pi4 pf pi1;
   plfi1=ci4 poin 2;plfi2=plfi1 plus (0. 0.);
   n=2;si ((i ega 6 ) ou (i ega 7));n=4;finsi;
   m = n / 2;
   ci1=cerc  n pi1 pf pi2;
   ci2=cerc  n pi2 pf pi3;
   ci3=cerc  n pi3 pf pi4;
   ci41=cerc  m pi4 pf plfi1;
   ci42=cerc  m plfi2 pf pi1;
   sui1=cout cim1 ci1;
   sui2=cout cim2 ci2;
   sui3=cout cim3 ci3;
   sui41=cout cim41 ci41;
   sui42=cout cim42 ci42;
   elim 0.001 (sui42 et sui1);
   elim 0.001 (sui1 et sui2 et sui3 et sui41);
   lfi1=(cont sui41) comp plfim1 plfi1;
   lfi2=(cont sui42) comp plfim2 plfi2;
   lhi=pim2 d 1 pi2;
   cim1=ci1;cim2=ci2;cim3=ci3;cim41=ci41;cim42=ci42;
   plfim1=plfi1;plfim2=plfi2;pim2=pi2;
   sut1=sui42 et sui1;
   sut2=sui2 et sui3 et sui41;
   si (i ega 1);
     sutt1=sut1;sutt2=sut2;lf1=lfi1;lf2=lfi2;lh=lhi;
   sinon;
     sutt1=sutt1 et sut1;sutt2=sutt2 et sut2;
     lf1=lf1 et lfi1;lf2=lf2 et lfi2;lh=lh et lhi;
   finsi;
fin bbbb;
lg=p0 d 5 p3;ld=p2 d 5 p5;
lhc= p4 d 2 pi2;
lig1=lg et (p3 d 8 p4) et lhc;
lig1=lig1 et (inve ci1) et ci42;
lig1=lig1 et (plfi2 d 2 p12 d 5 p0);
su1=surf lig1 plan;obj1=su1 et sutt1;elim 0.001 obj1;
lig2=ld et (p5 d 8 p4) et lhc;
lig2=lig2 et ci2 et ci3 et ci41;
lig2=lig2 et (plfi1 d 2 p11 d 8 p2);
su2=surf lig2 plan;obj2=su2 et sutt2;elim 0.001 obj2;
perobj1 = inve ( cont sutt1);
perobj2 = inve ( cont sutt2);
lifis1 = perobj1 comp pf plfi2;
lifis2 = perobj2 comp pf plfi1;

obj=obj1 et obj2;

*-----------DEFINITION DU MATERIAU-------------------;
objaf = modl obj MECANIQUE ELASTIQUE ISOTROPE;
mat=mate objaf youn 20000 nu 0.3;
*-----------MATRICE DE RIGIDITE----------------------;
rig=rigi mat objaf;
*-----------DEFINITION DU CHARGEMENT-----------------;
cdl=(bloq ux uy p0) et (bloq uy p2);
f1=pres mass objaf lg -10.;
f2=pres mass objaf ld -10.;
*-----------RESOLUTION ET CONTRAINTES----------------;
u=reso (rig et cdl) (f1 et f2);
sig=sigm mat objaf u;
*----CALCUL DU FACTEUR D'INTENSITE DE CONTRAINTES----;
*----APPEL DE LA PROCEDURE SIF  ---------------------;
SUPTAB = TABLE;
SUPTAB.'MODMIXTE' = VRAI;
SUPTAB.'LIFIS1' = lifis2;
SUPTAB.'LIFIS2' = lifis1;
SUPTAB.'FRTFISS' = pf;

SIF SUPTAB MAT U;
K1MOY = SUPTAB.K1;
K2MOY = SUPTAB.K2;

*-----DEPOUILLEMENT  --------------------------------;


mess '**********************';
mess '* SOLUTION ANALYTIQUE *';
mess '**********************';
mess 'K1th = 150.37 ';
EC1 = (K1MOY - 150.37) / K1MOY * 100;
mess 'ECART entre K1MOY et K1TH : 'EC1'%';
mess 'K2th =  -71.4 ';
EC2 = (K2MOY + 71.4) / K2MOY * 100;
mess'ECART entre K2MOY et K2TH : 'EC2'%';

*---------------------------------------------------;
*------------------MESSAGE D'ERREUR------------------*
SI ((EC1 < 0.6) ET (EC2 < 0.25));
  MESS 'ERR   0';
  ERRE  0;
SINON;
  MESS 'ERR   5';
  ERRE  5;
FINSI;
*---------------------------------------------------*
fin;
```

6.8 Test rupt8 Description sheet

Test name rupt8

Calculation type MECHANICS ELASTICITY

Finite element type TRI3 MODE TRID

Topic STRESS INTENSITY FACTOR FOR A CENTRAL CRACK IN A FINITE WIDTH PLATE SUBJECTED TO UNIFORM TENSILE STRESS

The structure is a solid plate enclosing a central crack . This structure is subject to a uniform load by a simple tensile stress test. Due to symmerty only 1/4 the structure is analysed. The CASTEM solution for the stress intensity factors K is given by the relationship :

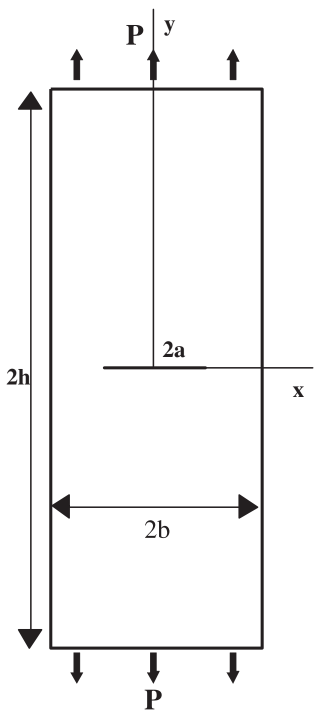
$K_I = (\sqrt{EG/1-\nu^2})$ *G is the energy release rate*

Finally this CASTEM K value for the stress intensity factor is compared to the result obtained with the analytical formula of ROOKE and CARTWRIGHT using the relationship :

$$\frac{K_I}{K_O} = \frac{1 - 0.5\frac{a}{b} + 0.326(\frac{a}{b})^2}{(1-\frac{a}{b})^{1/2}}$$

$$K_0 = \sigma\sqrt{\pi}a$$

Reference CASTEM Test NAFEMS : D.P.Rooke and D. J. Cartwright in Copendium of Intensity Factors HMSO (1976) : rupt8 Modélisation des structures élastiques dans CASTEM 2000. **Model description**



Central Crack in a finite width plate
subjected to uniform bending moment

GEOMETRY :

Width of the plate : 2b = 2 m
Length of the plate : 2h = 10.5 m
Length of the crack : 2a = 0.6 m
Thickness of the plate : e = 0.001m

MATERIAL CHARACTERISTICS :

E = 20000 daN mm^(-2)
nu = 0.3

LOADING CONDITIONS :

Uniform tensile stress
P = 5E6N m^(-2)

Test rupt8 **Results**

RESULTS

	$K_I(Nm^{-3/2})$
Reference	$50.1E5$
Castem	$50.8E5$

CASTEM FIGURES

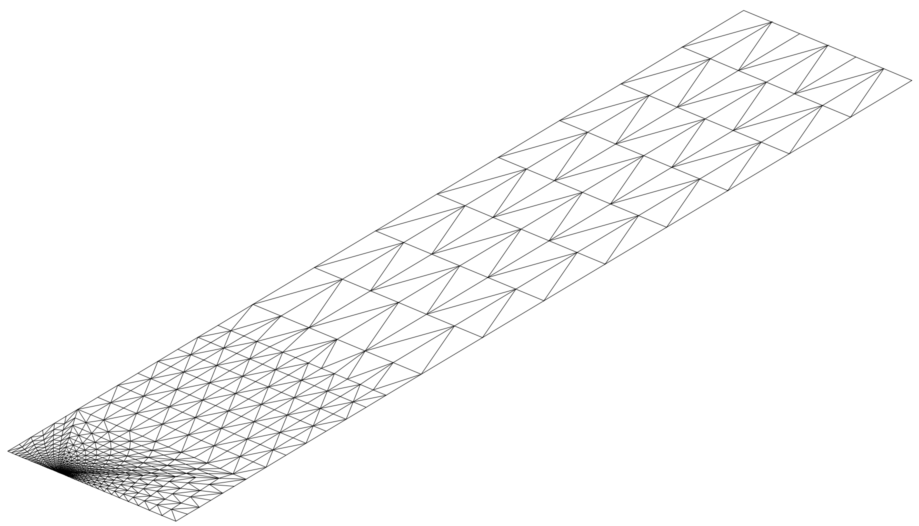


PLATE WITH CENTRAL CRACK SUBJECTED TO TENSILE STRESS

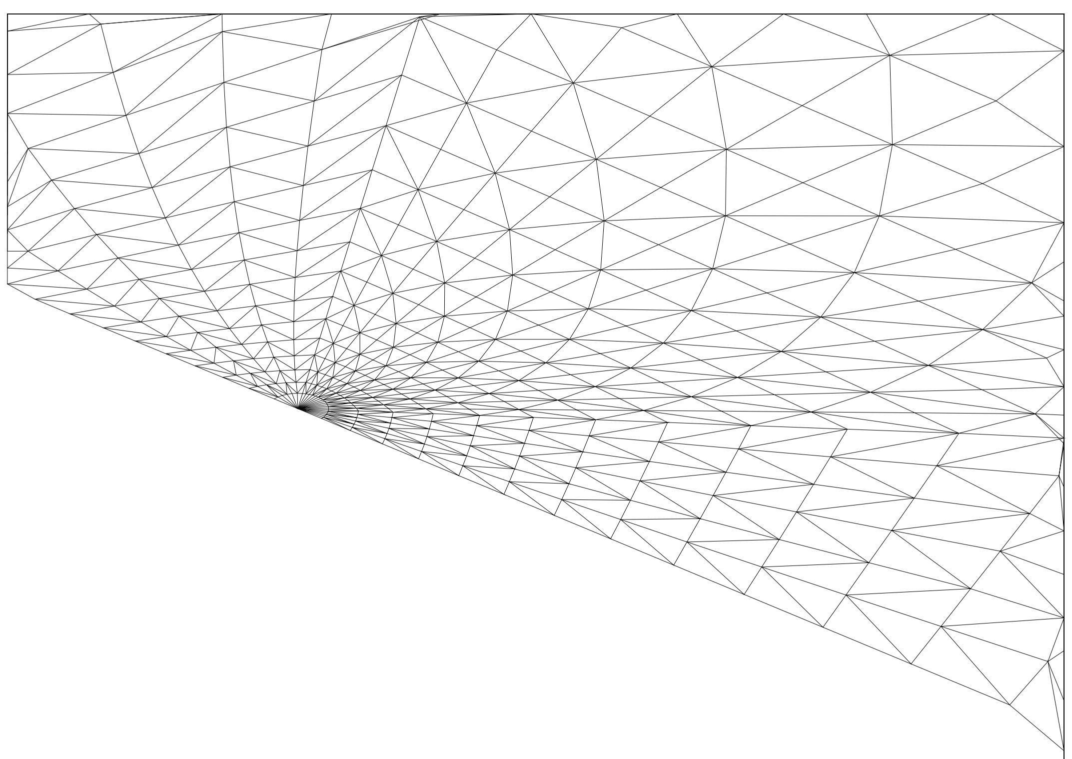


PLATE WITH CENTRAL CRACK SUBJECTED TO TENSILE STRESS

```
*          Test Rupt8.dgibi: Jeux de données
*          ---------------------------------
*
*****************************************************
*                                                   *
*     VALIDATION DE LA METHODE G_THETA EN CAS       *
*     D'UNE PLAQUE EN TRACTION PURE. SOLUTION       *
*  DE REFERENCE : Compendium of STRESS INTENSITY    *
*  FACTORS by Rooke & Cartwright. Section Plates    *
*   and Shells "Central crack in a finite width     *
*         plate : uniform bending moment"           *
*                                                   *
*                                                   *
*  GEOMETRIE :                                      *
*                                                   *
*  longueur de la fissure : 2a = 2 x 0.30 M         *
*  longueur de la plaque  : 2b = 2 x 1.00 M         *
*  Hauteur de la plaque   : 2h = 2 x 5.25 M         *
*  Epaisseur de la plaque : e  = 0.001    M         *
*                                                   *
*  MATERIAU :                                       *
*                                                   *
*  Module d'Young         : E = 2.E11               *
*  Coefficient de poisson : NU = 0.3                *
*                                                   *
*  CHARGEMENT :                                     *
*                                                   *
*  Traction pure          : SIG = 5.E6 N/M2         *
*                                                   *
*  SOLUTION ANALYTIQUE :                            *
*                                                   *
*  Le taux de restitution d'energie : G = 129.5     *
*                                                   *
*****************************************************
opti echo 0;
OPTI DIME 3 ELEM TRI3 DENS 1.;
N4 = 6; N8 = 13;
N12 = (N4 + N8)*(-1);
O1 = 0. 0. 1000.;
ADIM = 0.3; BDIM = 1.; TAFIS = ADIM / 33.;
HH1 = BDIM / 2.0;
P1 = 0. 0. 0.; P2 = BDIM 0. 0.; P3 = ADIM 0. 0.;
P3G = (ADIM - TAFIS) 0. 0.; P3D = (ADIM + TAFIS) 0. 0.;
ALF1 = ATG HH1 ADIM; ALF2 = ATG HH1 (BDIM - ADIM);
X1 = TAFIS * (COS ALF1); Y1 = TAFIS * (SIN ALF1);
P31 = (ADIM - X1) Y1 0.;
X1 = TAFIS * (COS ALF2); Y1 = TAFIS * (SIN ALF2);
P32 = (ADIM + X1) Y1 0.;
C1 = C N4 P3G P3 P31; C2 = C N8 P31 P3 P32;
C3 = C N4 P32 P3 P3D;
SU1 = COUT (C1 ET C2 ET C3) P3;
P4 = 0. HH1 0.; P5 = BDIM HH1 0.;
L1 = D N4 P1 P4 DINI (2.0*TAFIS) DFIN (8.0*TAFIS);
L2 = D N12 P4 P31 DINI (9.0*TAFIS) DFIN (0.3*TAFIS);
L3 = D N12 P1 P3G DINI (3.0*TAFIS) DFIN (1.5*TAFIS);
L4 = D N8 P4 P5 ;
L5 = D N12 P5 P32 DINI (14.0*TAFIS) DFIN (0.3*TAFIS);
L6 = D N4 P2 P5 DINI (2.0*TAFIS) DFIN (10.0*TAFIS);
L7 = D N12 P2 P3D DINI (10.0*TAFIS) DFIN (1.5*TAFIS);
SU2 = DALL L1 L3 C1 L2;
SU3 = DALL L2 L4 L5 C2;
SU4 = DALL L6 L5 C3 L7;
SU5 = L4 TRAN 7 (0. (1.0*BDIM) 0.);
L8 = COTE 3 SU5; Y1 = COOR 2 (POINT L8 FINA);
P6 = 0. (Y1 + (BDIM/4.)) 0.;
P7 =BDIM (Y1 + (BDIM/4.)) 0.;
L9 = D N4 P7 P6;
SU6 = COUT L8 L9;
* SU7 = L9 TRAN 8 (0. (3.5*BDIM) 0.);
* L10 = COTE 3 SU7; Y1 = COOR 2 (POINT L10 FINA);
* L11 = D 1 P3 P3D;
  SUT = COUL TURQ (SU1 ET SU2 ET SU3 ET SU4 ET SU5 ET
                   SU6 ET SU7);
  ELIM 1.E-8 SUT;
  L1 = (CONT SUT) ELEM APPU (SUT POIN
       DROI (0. 0. 0.) (BDIM 0. 0.) 1.E-8);
  L2 = (CONT SUT) ELEM APPU (SUT POIN
       DROI (BDIM 0. 0.) (BDIM Y1 0.) 1.E-8);
  L3 = (CONT SUT) ELEM APPU (SUT POIN
       DROI (BDIM Y1 0.) (0. Y1 0.) 1.E-8);
  L4 = (CONT SUT) ELEM APPU (SUT POIN
       DROI (0. Y1 0.) (0. 0. 0.) 1.E-8);
  PINI3 = POIN L3 INIT; PFIN3 = POIN L3 FINA;
  PMIL3 = (CHAN POI1 L3) DIFF (PINI3 ET PFIN3);
  MOEXT = 1. / ((NBNO L3) - 1);
  MOMIL = MOEXT * (NBNO PMIL3);
  ******
  ****** SOLUTION CONTRAINTE ET DEPLACEMENT
  ******
  EPAI_T = 0.001;
  NB_MUL = 21;
  EPAI_C = EPAI_T * 1.E-6;
  EPAI_I = (EPAI_T - (EPAI_C*NB_MUL))/(NB_MUL - 1);
  ELEFIS = SUT ELEM APPU LARG P3;
  ELEFIS = SUT ELEM APPU LARG ELEFIS;
  ELEFIS = SUT ELEM APPU LARG ELEFIS;
  TBEPAI = TABLE; TBEXCE = TABLE;
  REPETER BC_MUL1 ((2*NB_MUL) - 1);
    III = (&BC_MUL1 - NB_MUL) / 2.;
    TBEXCE.&BC_MUL1 = (EPAI_I + EPAI_C)*III;
    SI (EGA 0. ((&BC_MUL1/2.) - (&BC_MUL1/2)) 1.E-10);
      TBEPAI.&BC_MUL1 = EPAI_I;
    SINON;
      TBEPAI.&BC_MUL1 = EPAI_C;
    FINSI;
  FIN BC_MUL1;
  REPETER BC_MUL2 ((2*NB_MUL) - 1);
    SI (EGA &BC_MUL2 1);
     MOD_MUL = MODE ELEFIS MECANIQUE ELASTIQUE
               DKT CONS (CHAI &BC_MUL2);
     MAT_MUL = MATE MOD_MUL YOUN 2.E11 NU 0.3
               EPAI TBEPAI.&BC_MUL2 EXCE TBEXCE.&BC_MUL2;
    SINON;
     MOD_MU1 = MODE ELEFIS MECANIQUE ELASTIQUE
               DKT CONS (CHAI &BC_MUL2);
     MAT_MU1 = MATE MOD_MU1 YOUN 2.E11 NU 0.3
               EPAI TBEPAI.&BC_MUL2 EXCE TBEXCE.&BC_MUL2;
     MOD_MUL = MOD_MUL ET MOD_MU1;
     MAT_MUL = MAT_MUL ET MAT_MU1;
    FINSI;
  FIN BC_MUL2;
  MOD_R = MODE (SUT DIFF ELEFIS) MECANIQUE ELASTIQUE DKT;
  MAT_R = MATE MOD_R YOUN 2.E11 NU 0.3 EPAI EPAI_T;
  MO1 = MOD_MUL ET MOD_R;
  MA1 = MAT_MUL ET MAT_R;
  BL1 = BLOQ UY RX  (L7 ET L11);
  BL2 = BLOQ UX RY L4;
  BL3 = BLOQ UZ RZ L3;
  RI = BL1 ET BL2 ET BL3 ET (RIGI MA1 MO1);
  FO2 = ((FORC FY MOEXT (PINI3 ET PFIN3)) ET
        (FORC FY MOMIL PMIL3))*5.e3;
  FO3 = FO2;
  DEPTO = RESO RI FO3 ;
  REATO = REAC DEPTO RI;
  SIGTO = SIGMA MA1 MO1 DEPTO ;
  ******************************
```

```
SUPTAB = TABLE ;
SUPTAB.'OBJECTIF' = MOT 'J';
SUPTAB.'LEVRE_SUPERIEURE' = comp (cont sut) p3 p1;
SUPTAB.'FRONT_FISSURE' = P3 ;
SUPTAB.'MODELE' = MO1;
SUPTAB.'CARACTERISTIQUES' = MA1;
SUPTAB.'SOLUTION_RESO' = DEPTO;
SUPTAB.'CHARGEMENTS_MECANIQUES' = FO3;
SUPTAB.'ELEMENT_MULTICOUCHE' = ELEFIS ELEM 1;
SUPTAB.'COUCHE' = 7;
G_THETA SUPTAB;
* RECUPERATION DU RESULTAT ET COMPARAISONS ELEMENT 25 ;
*                                                     ;
G   = SUPTAB.'RESULTATS'.'GLOBAL'                     ;
Kth = 50.1E5                                          ;
K = (2.E11 * G)**0.5                                  ;
mess '  '                                             ;
mess '  '                                             ;
mess 'taux de restitution energetique  G :' G         ;
mess 'solution calculee            Kcal :' K          ;
mess 'solution analytique          Kthe :' Kth        ;
*                   CODE FONCTIONNEMENT               ;
RESI = abs (( k - Kth ) / Kth )                       ;
SI (RESI <  5E-2)                                     ;
    ERRE  0                                           ;
SINO                                                  ;
    ERRE  5                                           ;
FINSI                                                 ;
*RECUPERATION DU RESULTAT ET COMPARAISONS ELEMENT 117 ;
*                                                     ;
SUPTAB.'ELEMENT_MULTICOUCHE' = ELEFIS ELEM 117;
G_THETA SUPTAB;
G   = SUPTAB.'RESULTATS'.'GLOBAL'                     ;
Kth = 50.1E5                                          ;
K = (2.E11 * G)**0.5                                  ;
mess '  '                                             ;
mess '  '                                             ;
mess 'taux de restitution energetique  G :' G         ;
mess 'solution calculee            Kcal :' K          ;
mess 'solution analytique          Kthe :' Kth        ;
*                   CODE FONCTIONNEMENT               ;
RESI = abs (( k - Kth ) / Kth )                       ;
SI (RESI <  5E-2)                                     ;
    ERRE  0                                           ;
SINO                                                  ;
    ERRE  5                                           ;
FINSI                                                 ;
*RECUPERATION DU RESULTAT ET COMPARAISONS ELEMENT 118 ;
*                                                     ;
SUPTAB.'ELEMENT_MULTICOUCHE' = ELEFIS ELEM 118;
G_THETA SUPTAB;
G   = SUPTAB.'RESULTATS'.'GLOBAL'                     ;
Kth = 50.1E5                                          ;
K = (2.E11 * G)**0.5                                  ;
mess '  '                                             ;
mess '  '                                             ;
mess 'taux de restitution energetique  G :' G         ;
mess 'solution calculee            Kcal :' K          ;
mess 'solution analytique          Kthe :' Kth        ;
*                   CODE FONCTIONNEMENT               ;
RESI = abs (( k - Kth ) / Kth )                       ;
SI (RESI <  5E-2)                                     ;
    ERRE  0                                           ;
SINO                                                  ;
    ERRE  5                                           ;
FINSI                                                 ;
FIN;
```

6.9 Test rupt9 Description sheet

Test name rupt9

Calculation type MECHANICS ELASTICITY

Finite element type QUA8 DIME 2

Topic VALIDATION OF THE G_THETA AND T_PITETA PROCEDURES FOR AN OBLIQUE EDGE CRACK IN A PLATE SUBJECTED TO UNIFORM TENSILE STRESS

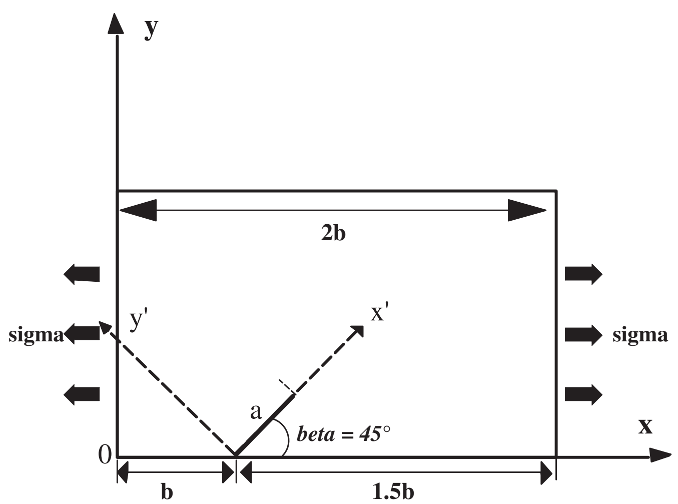
The structure is a finite plate enclosing an edge crack . This structure is subject to uniform tensile stress. The CASTEM solution for the stress intensity factor K is given with the displacement method in 3 nodes at the crack tip (including the mid-node) from the local displacements wich are computed in the local basis $(X'Y')$ for the crack by the relationship :

$$K_I = \sqrt{\frac{EG_I}{1-\nu^2}} \quad and \quad K_{II} = \sqrt{\frac{EG_{II}}{1-\nu^2}}$$

Finally this CASTEM K values for the stress intensity factor is compared to the results obtained with the analytical method .

Reference CASTEM Test NAFEMS : D.P.Rooke and D. J. Cartwright in Copendium of Intensity Factors HMSO (1976) : rupt7 Modélisation des structures élastiques dans CASTEM 2000. **Version** 97'

customer version **Model description**



Tension for a plate with the 45°angled kinked crack

GEOMETRY :
Width of the plate : b = 100 mm
Length of the plate : 2.5b = 250 mm
Length of the crack : a = 50 mm
beta = 45°

MATERIAL CHARACTERISTICS :
E = 20000 daN mm^(-2)
nu = 0.3

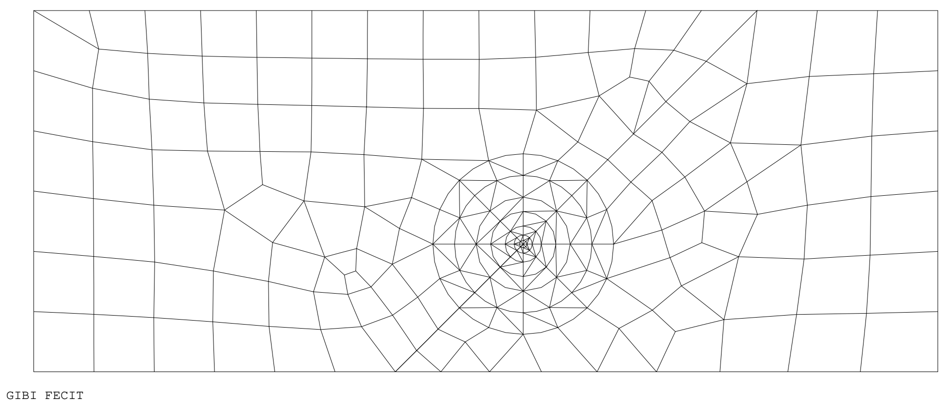
LOADING CONDITIONS :
Tensile Stress :
Sigma = 10 daN mm^(-2)

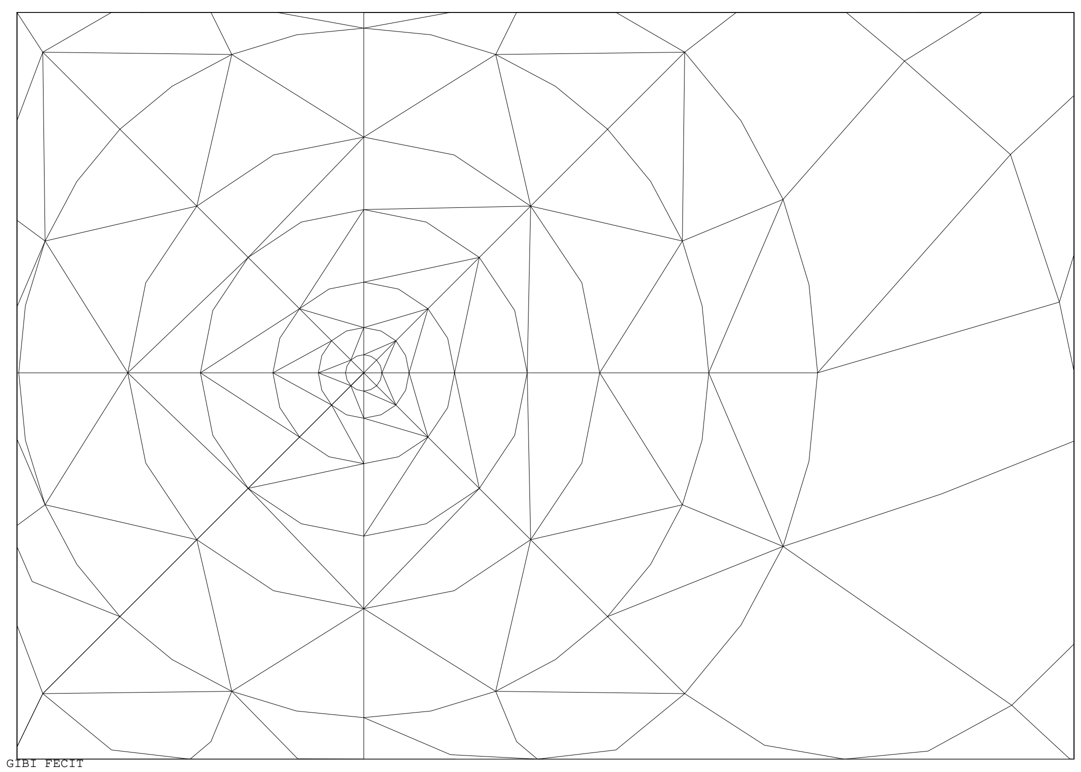
Test rupt9 **Results**

RESULTS

	$K_I(daNmm^{-3/2})$	$K_II(daNmm^{-3/2})$
Reference	149.77	-71.56
Castem	145.98	-72.28

CASTEM FIGURES





```
*          Test Rupt9.dgibi: Jeux de données
*          --------------------------------
*
*****************************************************
*                                                   *
*   VALIDATION DES PROCEDURE G_THETA ET T_PITETA    *
*     D'UNE PLAQUE EN TRACTION PURE. SOLUTION       *
*  DE REFERENCE : Compendium of STRESS INTENSITY    *
*  FACTORS by Rooke & Cartwright.                   *
*                                                   *
*                                                   *
*  GEOMETRIE :                                      *
*                                                   *
*  longueur de la plaque     : 250. MM              *
*  largeur de la plaque      : 100. MM              *
*  lonqueur de la fissure    : 50.  MM              *
*  Inclinaison de la fissure : 45.0 degrés          *
*                                                   *
*  MATERIAU :                                       *
*                                                   *
*  Module d'Young         : 2.E5 MPA                *
*  Coefficient de poisson : 0.3                     *
*                                                   *
*  CHARGEMENT :                                     *
*                                                   *
*  Traction pure          : 1.0 daN/MM**2           *
*                                                   *
*  SOLUTION ANALYTIQUE :                            *
*                                                   *
*  Les facteurs d'intensité de contraintes          *
*              K1 = 150.37 daN/mm**(-3/2)           *
*              K2 = -71.4 daN/mm**(-3/2)            *
*                                                   *
*****************************************************
'OPTION' 'DIME' 2 'ELEM' 'QUA8'
         'MODE' 'PLAN' 'DEFO'
         'ECHO'  1 ;
*** Erreur géométrique pour les éliminations
*** Plus petite que la fissure !
egeom = 1.D-6 ;
***
*** DD : Ouverture initiale de la fissure
***
DD = 1.D-3 ;
ddsur2 = DD '/' 2.0D0 ;
p11 = (125.D0 '-' ddsur2) 0.D0 ;
p12 = (125.D0 '+' ddsur2) 0.D0 ;
p0  = 25.D0  0.D0 ;
p2  = 275.D0 0.D0 ;
p3  = 25.D0  100.D0 ;
ppp = 125.D0 0.D0 ;
L = 50.D0 ;  theta = 45.D0 ;
cth = 'COS' theta ;
sth = 'SIN' theta ;
tth = sth '/' cth ;
pf1 = ppp 'PLUS' (L '*' (cth sth)) ;
p4  = ((100.D0 '/' tth) '+' 125.D0)  100.D0 ;
p5  = 275.D0 100.D0 ;

*******************************************************
*         CONSTRUCTION DU MAILLAGE                    *
*       AUTOUR DE LA POINTE DE FISSURE                *
*******************************************************
pd = 'PROG' 1.D0  2.5D0 5.D0  9.D0  13.D0 19.D0 25.D0 ;
cim11 = pf1 ;  cim2 = pf1 ;  cim3 = pf1 ;
cim12 = pf1 ;
plfim1 = pf1 ;  plfim2 = pf1 ;  pim3 = pf1 ;
*  *******************************************************
*  'REPETER' bbbb 7 ;
*    v = 'EXTRAIRE' pd &bbbb ;
     dd1 = (v '/' L) '*' ddsur2 ;
     xx1 = v '*' cth ;  mxx1 = (-1.0D0 '*' xx1) ;
     yy1 = v '*' sth ;
     xx2 = dd1 '*' sth '*' sth ;
     yy2 = dd1 '*' sth '*' cth ;  myy2 = (-1.0D0 '*' yy2);
     pi1 = pf1 'MOIN' (xx1  yy1) ;
     pi2 = pf1 'PLUS' (mxx1 yy1) ;
     pi3 = pf1 'PLUS' (xx1  yy1) ;
     pi4 = pf1 'MOIN' (mxx1 yy1) ;
     plfi1 = pi1 'MOIN' (xx2 myy2) ;
     plfi2 = pi1 'PLUS' (xx2 myy2) ;
     'SI' (&bbbb '>EG' 6) ;
       n = 4 ;
     'SINON'  ;
       n = 2 ;
     'FINSI' ;
   *******************************************************
     ci11 = 'CERCLE' n plfi1 pf1 pi2   ;
     ci2  = 'CERCLE' n pi2   pf1 pi3   ;
     ci3  = 'CERCLE' n pi3   pf1 pi4   ;
     ci12 = 'CERCLE' n pi4   pf1 plfi2 ;
     sui11 = 'COUTURE' cim11 ci11 ;
     sui2  = 'COUTURE' cim2  ci2  ;
     sui3  = 'COUTURE' cim3  ci3  ;
     sui12 = 'COUTURE' cim12 ci12 ;
     'ELIMINATION' egeom (sui11 'ET' sui2) ;
     'ELIMINATION' egeom (sui12 'ET' sui3) ;
   *******************************************************
     cim11 = ci11 ;  cim2 = ci2 ;  cim3 = ci3 ;
     cim12 = ci12 ;  plfim1 = plfi1 ;
     plfim2 = plfi2 ;  pim3 = pi3 ;
     sut1 = 'ET' sui11 sui2  ;
     sut2 = 'ET' sui3  sui12 ;
     'SI' (&bbbb 'EGA' 1) ;
       sutt1 = sut1 ;
       sutt2 = sut2 ;
     'SINON' ;
       sutt1 = 'ET' sutt1 sut1 ;
       sutt2 = 'ET' sutt2 sut2 ;
     'FINSI' ;
   'FIN' bbbb ;
   'OUBLIER' cim11  ;  'OUBLIER' cim2   ;  'OUBLIER' cim3;
   'OUBLIER' cim12  ;
   'OUBLIER' plfim1 ;  'OUBLIER' plfim2 ;  'OUBLIER' pim3;
   'MENAGE' ;

   *********************************
   *  CONSTRUCTION DU MAILLAGE TOTAL
   *********************************

   lg   = 'DROIT'  6  p0  p3 ;
   ld   = 'DROIT'  6  p2  p5 ;
   lhc  = 'DROIT' -5  p4  pi3 'DINI' 65 'DFIN' 50 ;
   lig1 = 'ET' lg   ('ET' ('DROIT' 13 p3 p4)  lhc) ;
   lig1 = 'ET' lig1 ('INVERSE' ('ET' ci2 ci11)) ;
   lig1 = 'ET' lig1 ('ET' ('DROIT' 3 plfi1 p11)
                          ('DROIT' 6 p11   p0)) ;
*  su1  = 'SURFACE' ('ORIENTER' lig1) 'PLANE' ;
*  obj1 = 'ET' su1 sutt1 ;
*
   lig2 = 'ET' ld   ('ET' ('DROIT' 3 p5 p4) lhc) ;
   lig2 = 'ET' lig2 ('ET' ci3 ci12) ;
   lig2 = 'ET' lig2 ('ET' ('DROIT'  3 plfi2 p12)
                    ('DROIT' -9 p12   p2 'DINI' 40
                                         'DFIN' 50)) ;
```

```
su2  = 'SURFACE' ('ORIENTER' lig2) 'PLANE' ;
obj2 = 'ET' su2 sutt2 ;
*
obj  = 'ET' obj1 obj2 ;
'ELIMINATION' egeom obj ;
cobj = 'ORIENTER' ('CONTOUR' obj) ;
*
* Levres de la fissure
*
ls = cobj 'COMPRIS' p11 pf1 ;
li = cobj 'COMPRIS' pf1 p12 ;
*-----------DEFINITION DU MATERIAU-------------------;
objmod = 'MODELISER' obj 'MECANIQUE' 'ELASTIQUE'
         'ISOTROPE' ;
objmat = 'MATERIAU'  objmod 'YOUN' 2.D5 'NU  ' 0.3D0 ;
*-----------MATRICE DE RIGIDITE----------------------;
rig = 'RIGIDITE' objmat objmod ;
*--------------CONDITIONS AUX LIMITES----------------;
cal = 'ET' ('BLOQUE' 'UX' 'UY' p0)
           ('BLOQUE'      'UY' p2) ;
rigtot = 'ET' rig cal ;
*-----------DEFINITION DU CHARGEMENT-----------------;
f1 = 'PRES' 'MASS' objmod -10.D0 lg ;
f2 = 'PRES' 'MASS' objmod -10.D0 ld ;
f0 = 'ET' f1 f2 ;
*-----------RESOLUTION ET CONTRAINTES----------------;
u0 = 'RESO' rigtot f0 ;
sig0 = 'SIGMA' objmod objmat u0 ;

**********************************************
** fabrication de déplacements THETA et PI ***
**********************************************

SUPTAB = 'TABLE' ;
SUPTAB . 'OBJECTIF' = 'MOT' 'DECOUPLAGE' ;
SUPTAB . 'LEVRE_SUPERIEURE' = ls ;
SUPTAB . 'LEVRE_INFERIEURE' = li ;
SUPTAB . 'FRONT_FISSURE' = pf1 ;
SUPTAB . 'MODELE' = objmod ;
SUPTAB . 'CARACTERISTIQUES' = objmat ;
SUPTAB . 'SOLUTION_RESO' = u0 ;
SUPTAB . 'CHARGEMENTS_MECANIQUES' = f0 ;
SUPTAB . 'COUCHE' = 6 ;
G_THETA SUPTAB ;

**********************************************
************ Affichage des résultats *********
**********************************************

K1 = SUPTAB . 'RESULTATS' . 'I' ;
K2 = SUPTAB . 'RESULTATS' . 'II' ;
'MESSAGE'  'K1 = ' K1 'K2 = ' K2 ;
ER1 = 'ABS' (((K1 '-' 150.37D0) '/' 150.37D0) '*'
      100.D0) ;
ER2 = 'ABS' (((K2 '+'   71.4D0) '/'   71.4D0) '*'
      100.D0) ;
'MESSAGE' 'ER1 = ' ER1 'ER2 = ' ER2 ;
'SI'  ((ER1 '>' 2.D0) 'OU' (ER2 '>' 2.D0)) ;
  'ERREUR' 5 ;
'SINON' ;
  'ERREUR' 0 ;
'FINSI' ;
'FIN' ;
```

Chapter 7

COMPOSITE

7.1 Test comp1 Description sheet

Test name comp1

Calculation type MECHANICS COMPOSITE 3D

Finite element type DKT

Topic
A two layer composite cylinder subjected to internal pressure.

The cylinder fibers are wound around the Z-axis ($\pm 45^o$). The cylinder is subjected to an internal pressure. The displacements of its bottom end are constrained along the Z axis.

In order to reduce the number of freedom degrees, one of the top nodes is forced to move along X and Y-axis and in rotation around Z-axis (node PB).

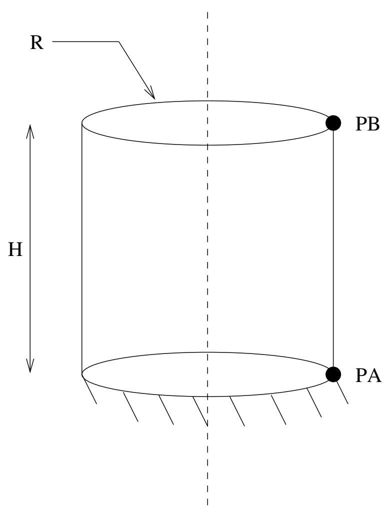
The pressure is perpendicular to the internal surface of the cylinder.

Goal Find the radial displacement of the point PA (1.05, 0, 0) belonging to the bottom of the cylinder.

Reference CASTEM Report CEA-DMT 89/184 : Modélisation des composites de type multi-couches dans CASTEM 2000.

Version 97' customer version

Model description



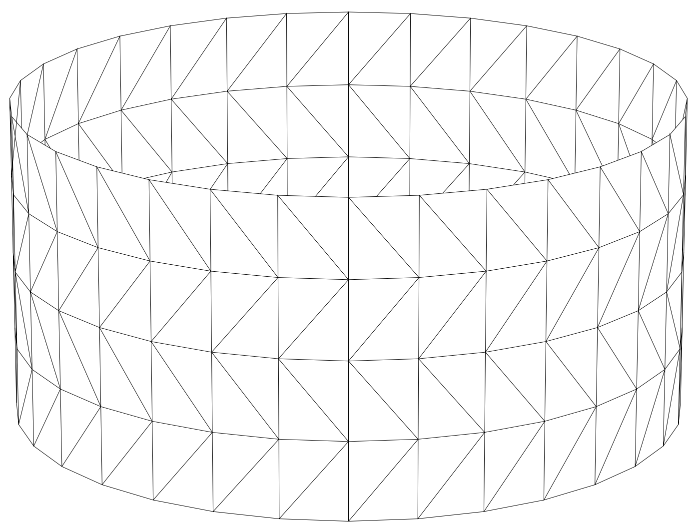
H	$= 1m$	E_1	$= 7.10^6 Pa$
R	$= 1.05m$	E_2	$= 1.3.10^6 Pa$
th_{tot}	$= 0.1m$	G_{12}	$= 5.10^5$
P	$= 100Pa$	ν_{12}	$= 0.28$

Test comp1

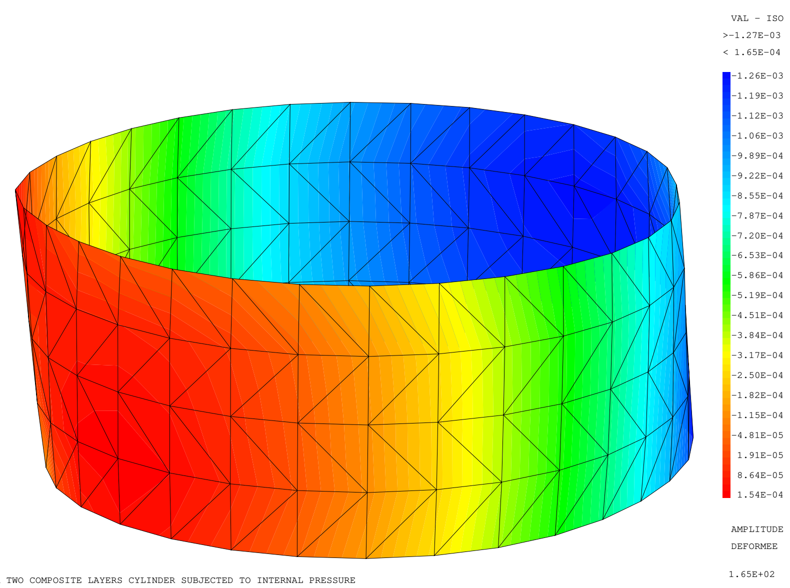
Results

	U_r
Theory	1.3776E-4 m
Castem	1.3777E-4 m

CASTEM figures



A TWO COMPOSITE LAYERS CYLINDER SUBJECTED TO INTERNAL PRESSURE



A TWO COMPOSITE LAYERS CYLINDER SUBJECTED TO INTERNAL PRESSURE

Results

```
*          Test Comp1.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
* SI GRAPH = N PAS DE GRAPHIQUE AFFICHE
* SINON SI GRAPH DIFFERENT DE N TOUS LES
* GRAPHIQUES SONT AFFICHES
*
GRAPH = 'N';
*
SAUT PAGE ;
*SI (NEG GRAPH 'N') ;
*  OPTION ECHO 1 ;
*SINO ;
*  OPTION ECHO 0 ;
*FINSI ;
*****************************************************
*               CYLINDRE COMPOSITE BICOUCHE
*          FIBRES ENROULEES -45/+45 AUTOUR DE L'AXE
*                    PRESSION INTERNE
*
*
* Un cylindre bloqué à sa base en déplacement suivant
* l'axe Z est soumis à une pression interne.
*
* Afin de réduire le nombre de degrés de liberté,
* un noeud du sommet est bloqué en translation suivant
* X et Y et en rotation suivant Z (noeud PB).
*
* La pression est normale à la surface interne
* du cylindre.
*
* Le déplacement radial d'un noeud de la base est
* comparé à celui que l'on obtiendrait par méthode
* analytique : 1.3776E-4
*
*****************************************************
TITRE
'CYLINDRE COMPOSITE BICOUCHE SOUS PRESSION INTERNE';
OPTION DIME 3 ELEM TRI3 MODE TRIDIM ;

TEMPS ;
DENS 0.1 ;
*____________________________________________________
*   GEOMETRIE
*____________________________________________________
R = 1.05 ; H = 1. ;
PA = R 0. 0.  ; PB = R 0. H ;
O1 = 0. 0. 0. ; O2 = 0. 0. H ;
NR1 = 1. 1. 0. ;
LI1 = PA D 4 PB ;
CYLP = ROTA LI1 8 90 O1 O2 ;
CYL1 = ORIENTER CYLP NR1 ;
CYL2 LIB = TOURNER CYL1 LI1 90 O1 O2 ;
CYL3 LIC = TOURNER CYL2 LIB 90 O1 O2 ;
CYL4 LID = TOURNER CYL3 LIC 90 O1 O2 ;
CYL = CYL1 ET CYL2 ET CYL3 ET CYL4 ;
ELIM CYL ;
OEIL = 10. 10. 5. ;
*
SI (NEG GRAPH 'N') ;
  TRACE OEIL CACH CYL ;
FINSI ;
*
MODORT = MODL CYL MECANIQUE ELASTIQUE ORTHOTROPE DKT ;
*

*____________________________________________________
*   DESCRIPTION DU MATERIAU ORTHOTROPE ET
*   CREATION DE LA RIGIDITE DE LA COUCHE 1
*____________________________________________________

MAT1 = MATR MODORT DIRE O2 INCL 45 NR1 YG1 7E6 YG2
       1.3E6 NU12 0.28 G12 5E5 ;
CAR1 = CARB MODORT EPAI 0.05 EXCENTREMENT 0.025 ;
MAT1=MAT1 ET CAR1;

RIG1 = RIGI MODORT MAT1  ;
*____________________________________________________
*   DESCRIPTION DU MATERIAU ORTHOTROPE ET
*   CREATION DE LA RIGIDITE DE LA COUCHE 2
*____________________________________________________
MAT2 = MATR MODORT DIRE O2 INCL -45 NR1 YG1 7E6 YG2
       1.3E6 NU12 0.28 G12 5E5 ;

CAR2 = CARB MODORT EPAI 0.05 EXCENTREMENT -0.025 ;
MAT2=MAT2 ET CAR2;
RIG2 = RIGI MODORT MAT2  ;
*____________________________________________________
*____________________________________________________
CO1 = COTE 2 CYL1 ; CO2 = COTE 2 CYL2 ;
CO3 = COTE 2 CYL3 ;
CO4 = COTE 2 CYL4 ;
COB = CO1 ET CO2 ET CO3 ET CO4 ;
CDL1 = BLOQ UZ COB ;
CDL2 = BLOQ UX UY RZ PB ;
CDL = CDL1 ET CDL2 ;
*____________________________________________________
*   CALCUL
*____________________________________________________
RIGITOT = RIG1 ET RIG2 ET CDL ;
FP = PRES COQUE MODORT 100. NORM ;
DEPL = RESO RIGITOT FP ;
*____________________________________________________
*____________________________________________________
DEFO1=DEFO CYL DEPL;
DX=EXCO DEPL UX;
SI (NEG GRAPH 'N') ;
 TRAC OEIL CACH DEFO1 DX;
FINSI;
*
D = EXTR DEPL UX PA ;
SAUT PAGE ;
MESS ' DEPLACEMENT RADIAL EN PA DE REFERENCE
:    1.37762E-4 ' ;
SAUT 1 LIGN ;
MESS ' DEPLACEMENT RADIAL EN PA CALCULE
:   ' D ;
SAUT 2 LIGN ;
TEMPS ;
*                    CODE BON FONCTIONNEMENT
DEPREF=1.37762E-4;
RESI=ABS((D-DEPREF)/DEPREF);
SI (RESI <EG 1E-2);
   ERRE 0 ;
SINO;
   ERRE 5 ;
FINSI ;
FIN;
```

Test comp1 **Comments**

1. **Description of orthotropic material**

   MAT1 = MATER MODORT1 DIRE O2 INCL 45 NR1 YG1 7E6 YG2 1.3E6 NU12 0.28 G12 5E5 ;

   The orthotropy is defined by the first orthotropic direction. First, a reference vector VEC1 is defined, thus giving the first orthotropic direction.

   - Reference vector VEC1

   DIRE O2 : projecting the direction O2 on the tangential plane to the shell.

   - First orthotropic direction

   INCL 45 NR1 : The first orthotropic direction makes an angle of 45^o with the reference direction VEC1. The direction NR1 gives the direction perpendicular to the shell, orientated outwards.

2. **Definition of the offsetting with the median line**

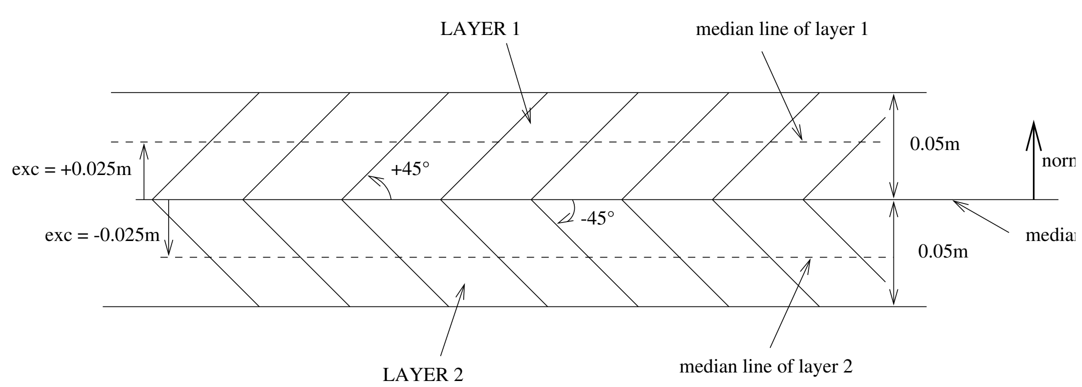
   CAR1 = CARAC MODORT1 EPAI 0.05 EXCENTR 0.025 ;

   The median line of the layer 1 is 0.025 meters (=EPAI/2) away from the median line of the cylinder (R = 1.05).

3. **Drawing of displacements with deformed shape**

   TRAC OEIL CACH DEFO1 DX;

   It is possible to plot the isodisplacements or isostrains on the deformed shape. For this, the object DEFO1 (DEFORME type) must be added in the TRAC operator.



7.2 Test comp2 Description sheet

Test comp2 **Description sheet**

Test name comp2

Calculation type MECHANICS COMPOSITE 3D

Finite element type DKT

Topic
A two layer composite shell subjected to a distributed pressure.

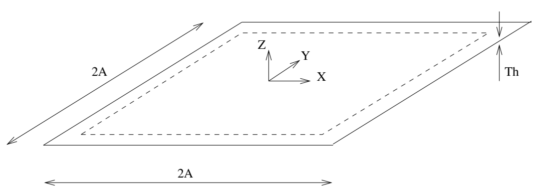
The shell fibers make an angle of $\pm 45^o$ with the median line.
The square shell is supported on its circumference (displacements and rotation along Z-axis are constrained).
The middle node only can move along the vertical Z-axis.
The pressure is distributed on the entire shell (upper face).

Goal Find the axial displacement of the middle point PO (0, 0, 0).

Reference CASTEM Report CEA-DMT 89/186 :

Version 97' customer version

Model description



$$
\begin{array}{ll}
A & = 127mm \\
Th_{tot} & = 5.08mm \\
P & = 0.6894MPa
\end{array}
\qquad
\begin{array}{ll}
E_1 & = 276.10^3 MPa \\
E_2 & = 6.3.10^3 MPa \\
G_{12} & = 3.4.10^3 \\
\nu_{12} & = 0.25
\end{array}
$$

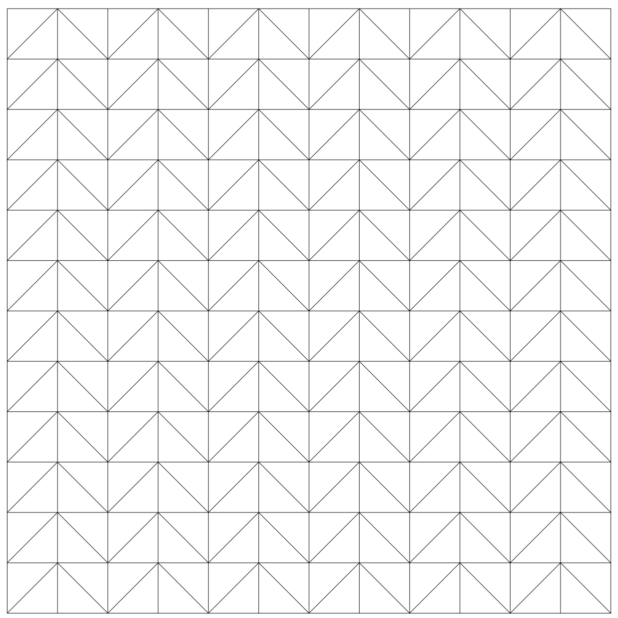
Test comp2

Results

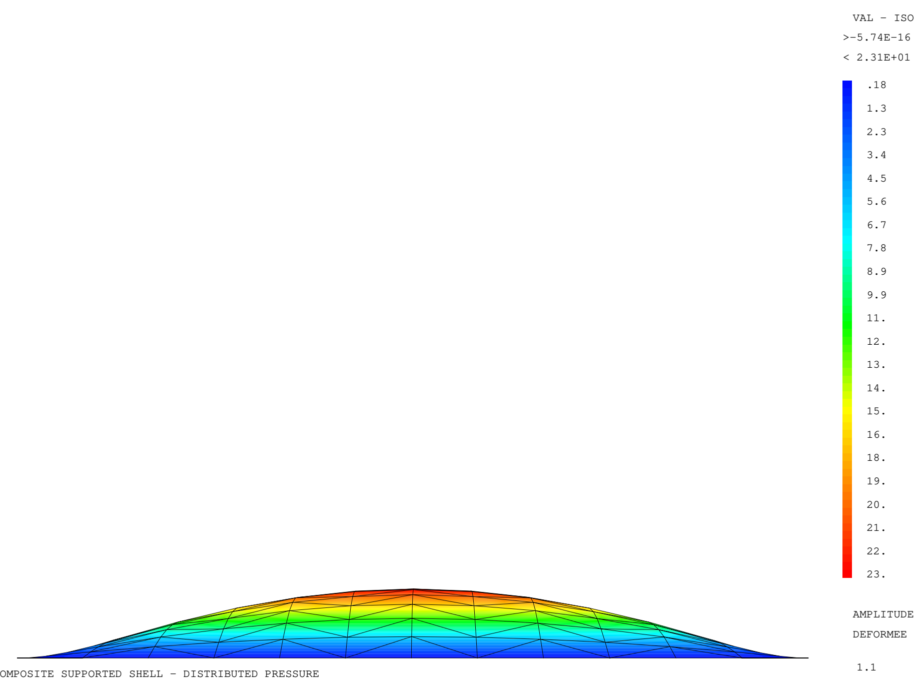
Results

	$U_z(PO)$
Reference	23.25
Castem	23.126

CASTEM figures



COMPOSITE SUPPORTED SHELL - DISTRIBUTED PRESSURE



COMPOSITE SUPPORTED SHELL - DISTRIBUTED PRESSURE

```
*          Test Comp2.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTION ECHO 0 ;
* SI GRAPH = N PAS DE GRAPHIQUE AFFICHE
* SINON SI GRAPH DIFFERENT DE N TOUS
* LES GRAPHIQUES SONT AFFICHES
*
GRAPH = 'N' ;
*
SAUT PAGE;
SI (NEG GRAPH 'N') ;
  OPTION ECHO 1 ;
SINO ;
  OPTION ECHO 0 ;
FINSI ;
*****************************************************
*                PLAQUE BI-COUCHE A +/- 45
*               APPUYEE , PRESSION REPARTIE
*                       ELEMENT DKT
*
*
* Une plaque composite carrée est appuyée sur toute sa
* périphérie (blocage en déplacement suivant Z et en
* rotation suivant Z).
*
* Le noeud situé au centre de la plaque ne peut se
* déplacer que suivant l'axe vertical Z (blocage en
* déplacement et en rotation suivant Z).
*
* Une pression répartie est appliquée sur toute
* la plaque.
*
* Le déplacement calculé au centre de la plaque est
* comparé au résultat théorique : 23.25
*
*****************************************************
*
SAUT PAGE ;
TITRE 'PLAQUE COMPOSITE APPUYEE - PRESSION REPARTIE';
OPTI DIME 3 ELEM TRI3 ECHO 0 ;
*____________________________________________________
*  GEOMETRIE
*____________________________________________________

TEMPS ;

A = 127. ; B = A ; AM = -127. ; BM = AM ;
H = 5.08 ; ES = H/4 ; EI = H/-4 ; S = H/2 ;
P = 0.6894 ;
N = 12 ;
P0 = 0. 0. 0. ;
P1 = AM BM 0. ; P2 = A BM 0. ;
P3 = A B 0.   ; P4 = AM B 0. ;
V1 = 0. (2*B) 0. ; V2 = 0. 0. 1. ;
L1 = DROI N P1 P2 ;

SI1=L1 TRAN V1 N;
S1=ORIE SI1 V2;
CONF (S1 POIN PROC P1) P1 ;
CONF (S1 POIN PROC P2) P2 ;
CONF (S1 POIN PROC P3) P3 ;
CONF (S1 POIN PROC P4) P4 ;
CONF (S1 POIN PROC P0) P0 ;
*
SI (NEG GRAPH 'N') ;
  TRACE S1 V2 ;
FINSI ;
*____________________________________________________
*  MATERIAUX
*____________________________________________________
MOD1 = MODL S1 MECANIQUE ELASTIQUE ORTHOTROPE DKT;

CARS = CARB MOD1 EPAI S EXCE ES ;
CARI = CARB MOD1 EPAI S EXCE EI ;
E11 = 276E3 ; E22 = 6.9E3 ; N12 = 0.25 ; GX = 3.4E3 ;
MATS =MATR MOD1 DIRE V1 INCLINE 45. V2 YG1 E11 YG2 E22
           NU12 N12 G12 GX ;
MATI=MATR MOD1 DIRE V1 INCLINE -45. V2 YG1 E11 YG2 E22
          NU12 N12 G12 GX ;
MATS=MATS ET CARS;
MATI=MATI ET CARI;
*____________________________________________________
*  CONDITIONS AUX LIMITES ET RIGIDITE
*____________________________________________________
RIS = RIGI MOD1 MATS  ;
RII = RIGI MOD1 MATI  ;
CL=(BLOQ UZ (CONT S1)) ET (BLOQ RZ (CONT S1)) ET
   (BLOQ UX UY RX RY P0);
RIT = RIS ET RII ET CL ;
*____________________________________________________
*  CALCUL ET SORTIE
*____________________________________________________
FP = PRES COQU MOD1 P V2 ;
DEP = RESO RIT FP ;
FP1 = EXTR DEP UZ P0 ;
SAUT PAGE ;
SU1=DEFO S1 DEP;
DZ=EXCO DEP UZ;
OEIL= 0 1000 0;
SI (NEG GRAPH 'N') ;
  TRAC OEIL CACH SU1 DZ;
FINSI ;
*
MESS 'FLECHE THEORIQUE      23.25' ;
SAUT 1 LIGN ;
MESS 'FLECHE CALCULEE           ' FP1 ;
SAUT 2 LIGN ;
TEMPS ;
*          CODE FONCTIONNEMENT
FLEREF = 23.25;
RESI = ABS ((FP1-FLEREF)/FLEREF);
SI (RESI <EG 1E-2);
    ERRE 0 ;
SINO;
    ERRE 5 ;
FINSI;
FIN ;
```

Chapter 8

CONCRETE

8.1 Test beton **Description sheet**

Test name beton

Calculation type ELASTIC PLASTIC MECHANICS - CONCRETE **Finite element type** CUB8

Topic Tensile test on a cubic piece made of concrete.

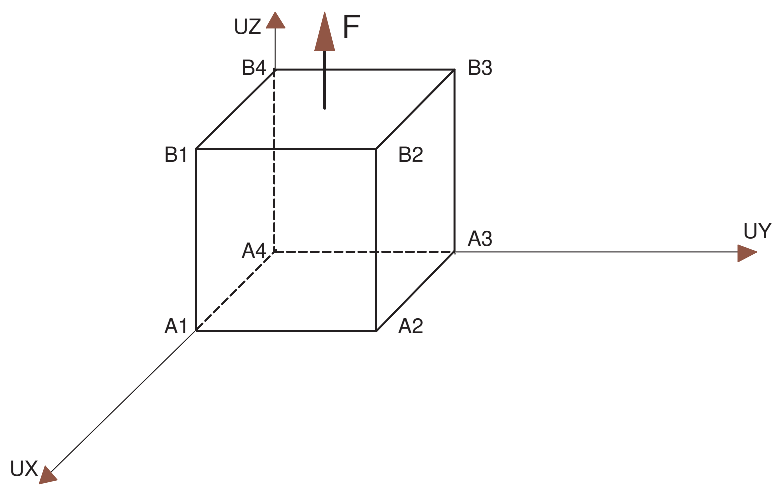
The structure is an embedded cube subjected to a tensile stress (imposed displacements). The structure follows a concrete elastic-plastic model. The limite in tension in the first direction for concrete (LRT1) is 4 MPa. **Goal** Test the model of concrete material for two new structures (MMODEL and MCHAMELEM).

Reference NAH (93/01/19)

Version 97' customer version

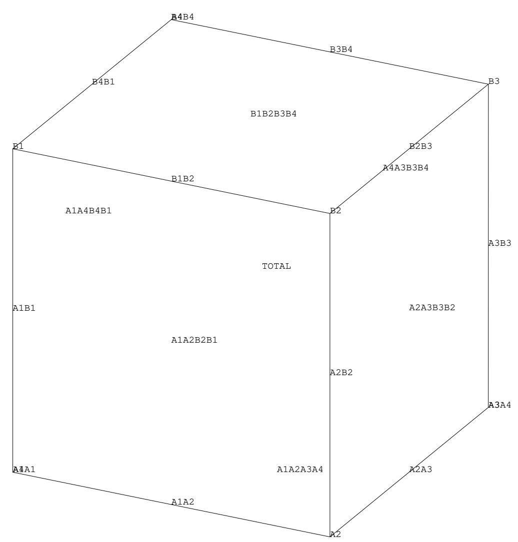
Model description

E	=	40000 MPa
NU	=	0.2
ALPH	=	$1.\ 10^{-5}$
LTR1	=	$4.MPa$

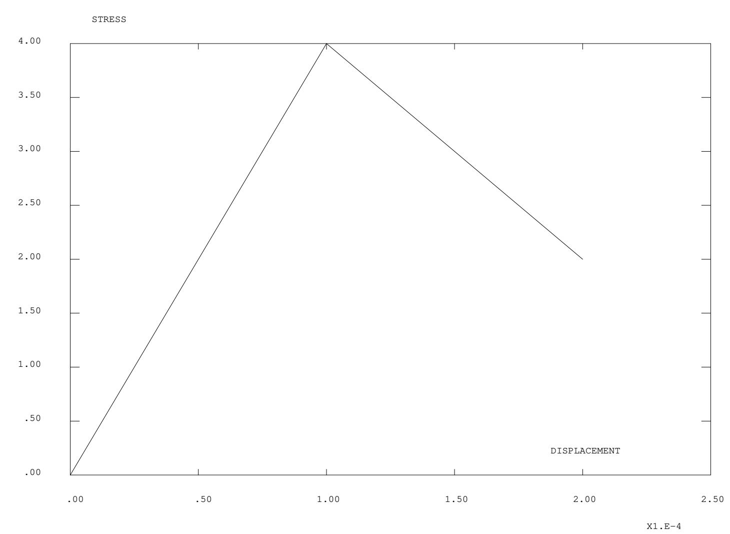


Test beton **Results**

CASTEM FIGURES



CONCRETE CUBE SUBJECTED TO TENSILE STRENGTH



CONCRETE CUBE SUBJECTED TO TENSILE STRENGTH

```
*          Test Beton.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
* CAS TEST DU 93/01/19     PROVENANCE : NAH
OPTION ECHO 1;
GRAPH='N';
SAUT PAGE ;
*
*---------------------------------------------------*
*                                                   *
*   Exemple simple de calcul mécanique élastique    *
*  plastique beton fonctionnant avec les nouvelles  *
*  structures MMODEL et MCHAMELEM                   *
*                                                   *
*  Un cube en béton soumis à un chargement de       *
*  traction uniaxial suivant l'axe OZ               *
*                                                   *
*  La limite en traction simple du béton LTR1 est   *
*  égale à 4 MPa                                    *
*                                                   *
*---------------------------------------------------*
* Définition des options
*
          OPTI DIME 3 ELEM CUB8 MODEL TRIDIM ;
*opti impi 9 ;
N = 1 ;
*
* Maillage d'un cube de N*N*N éléments CUB8
*
DENS N ;
*
A1 = 0 0 0 ;
A2 = N 0 0 ;
A3 = N N 0 ;
A4 = 0 N 0 ;
B1 = 0 0 N ;
B2 = N 0 N ;
B3 = N N N ;
B4 = 0 N N ;
*
A1A2 = A1 D N A2 ;
A2A3 = A2 D N A3 ;
A3A4 = A3 D N A4 ;
A4A1 = A4 D N A1 ;
B1B2 = B1 D N B2 ;
B2B3 = B2 D N B3 ;
B3B4 = B3 D N B4 ;
B4B1 = B4 D N B1 ;
A1B1 = A1 D N B1 ;
A2B2 = A2 D N B2 ;
A3B3 = A3 D N B3 ;
A4B4 = A4 D N B4 ;
*
A1A2A3A4 = A1A2 A2A3 A3A4 A4A1 DALLE PLAN ;
B1B2B3B4 = B1B2 B2B3 B3B4 B4B1 DALLE PLAN ;
*
A1A2B2B1 =       A1A2  A2B2 (INVE B1B2) (INVE A1B1)
                 DALLE PLAN ;
A4A3B3B4 = (INVE A3A4) A3B3       B3B4  (INVE A4B4)
           DALLE PLAN ;
*
A2A3B3B2 =       A2A3  A3B3       B2B3  (INVE A2B2)
                 DALLE PLAN ;
A1A4B4B1 = (INVE A4A1) A4B4       B4B1  (INVE A1B1)
           DALLE PLAN ;
*
TOTAL = PAVE A1A2A3A4 B1B2B3B4 A1A2B2B1 A4A3B3B4
             A2A3B3B2 A1A4B4B1 ;
*
*TRAC TOTAL CACH (1000 -2000 1000) ;
*
*
* Formulation, matériau défini à l'aide des
* opérateurs MODL et MATR
*
      MO = MODL TOTAL MECANIQUE ELASTIQUE PLASTIQUE
           BETON CUB8 ;
      MA = MATR MO YOUN 40000 NU 0.20 ALPH 1E-5
                          LTR1 4 ;
*
* Conditions de blocages
*
                        ENC1 = BLOQ UX A1A4B4B1 ;
                        ENC2 = BLOQ UY A1A2B2B1 ;
                        ENC3 = BLOQ UZ A1A2A3A4 ;
                        ENC = ENC1 ET ENC2 ET ENC3 ;
*
* Blocage pour déplacements imposés
*
*                       END1 = BLOQ UX A2A3B3B2 ;
*                       END2 = BLOQ UY A4A3B3B4 ;
                        END3 = BLOQ UZ B1B2B3B4 ;
*                       END = END1 ET END2 ET END3 ;
                        END = END3 ;
                        EN = ENC ET END ;
*
*
* Valeur des déplacements imposés
*
*                     FEXT1 = DEPI END1 1E-4  ;
*                     FEXT2 = DEPI END2 1E-4  ;
                      FEXT3 = DEPI END3 1E-4  ;
*                     FEXT = FEXT1 ET FEXT2 ET FEXT3 ;
                      FEXT = FEXT3 ;
*
* Force en fonction du temps
*
 lt1 = prog 0.  10. ;
 lt2 = prog 0.  10. ;
 ev  = evol rouge manu temps lt1 force lt2 ;
 chat  = char DIMP fext ev ;

*
* Préparation à nonlin
*
ltt = prog 0 1 2 ;
tab1 = table;
tab1.blocages_mecaniques = EN;
tab1.caracteristiques = ma;
tab1.modele = mo;
tab1.chargement = chat;
tab1.temps_calcules = ltt;
pasapas tab1;
*
* Traitement des résultats
*
 ptt = prog 0 ;
 psz = prog 0 ;
   ic = 0 ;
 repeter mabou 2 ;
   ic = ic + 1 ;
   dd = tab1 . deplacements . ic;
   ptt = ptt et ( prog ( extr dd uz b3 ) ) ;
   ss = tab1.contraintes.ic ;
   sz = extr ( chan chpo mo ss ) smzz b3 ;
   psz = psz et ( prog sz ) ;
```

```
 fin mabou ;
*
evsz = evol vert manu deplacement ptt contrainte  psz ;
*dess evsz ;
*
* Code de fonctionnement
*
si ( (abs ( sz - 2. )) <  1.E-6  ) ;
   erre 0 ;
sino ;
   erre 5 ;
finsi ;
*
fin ;
```

Test beton **Comments**

- The BETON model in plane strain, axisymmetrical and 3D

```
MO = MODE TOTAL MECANIQUE ELASTIQUE PLASTIQUE
          BETON CUB8 ;
MA = MATE MO YOUN 40000 NU 0.20 ALPH 1E-5 LTR1 4 ;
```

In this model, the concrete behavior is non-linear in tension, and linear everywhere else.
('LTR1') : limit in tension in the 1st direction (YOUN*1.2E-4 by default)
('ETR1') : failure strain in tension in the 1st direction (3*LTR1/YOUN by default)
('LTT1') : transitional limit in tension in the 1st direction (0. by default)
('ETT1') : deformation corresponding to LTT1 (ETR1 by default)
('ERS1') : residual deformation in tension in the 1st direction (0. by default)
('VF1i') : three components of the VF1 vector specifying the direction i connected with LTR1

('LTR2') : limit in tension in the 2nd direction (by default LTR1)
('ETR2') : failure strain in tension in the 2nd direction (3*LTR2/YOUN by default)
('LTT2') : transitional limit in tension in the 2nd direction (0. by default)
('ETT2') : deformation corresponding to LTT2 (ETR2 by default)
('ERS2') : residual deformation in tension in the 2nd direction (0. by default)
('VF2i') : three components of the VF2 vector specifying the direction i connected with LTR2

('LTR3') : limit in tension in the 3rd direction (LTR1 by default)
('ETR3') : failure strain in tension in the 3rd direction (3*LTR3/YOUN by default)
('LTT3') : transitional limit in tension in the 3rd direction (0. by default)
('ETT3') : deformation corresponding to LTT3 (ETR3 by default)
('ERS3') : residual deformation in tension in the 3rd direction (ERS1 by default)
('VF3i') : three components of the VF3 vector specifying the direction i connected with LTR3, required only in 3D when necessary.

('BETR') : residual reduction coefficient for the shear modulus in case of cracking (between 0. and 1., 0.1 by default)
Caution : The VF1, VF2 and VF3 vectors must be orthogonal.
For computation with a tension limit different from the two others, it is required that the vector corresponding to this limit VF1, VF2 or VF3 be defined.

When LTR1, LTR2 and LTR3 are given, both the VF1 and VF2 vectors are required to define the directions 1, 2 and 3.

Non-zero values corresponding to initial cracks openings in the directions 1, 2 and 3 can be introduced by means of the TAB1 table used in the NONLIN procedure through TAB1.'VARI'.'OUV1', TAB1.'VARI'.'OUV2', TAB1.'VARI'.'OUV3' may be input.

8.2 Test precont1 Description sheet

Test name precont1

Calculation type REINFORCED ELASTIC MECHANICS - BARR - 2D **Finite element type** QUA4

Topic Calculation with passive reinforcement

The structure is a reinforced concrete beam. The reinforcement is made of tendon. The beam is subjected to pressure on upper surface, and embedded on its lower side. **Goal** Test the KABL, EQUI RIGI and PROI

ARMA operators with passive reinforcement.

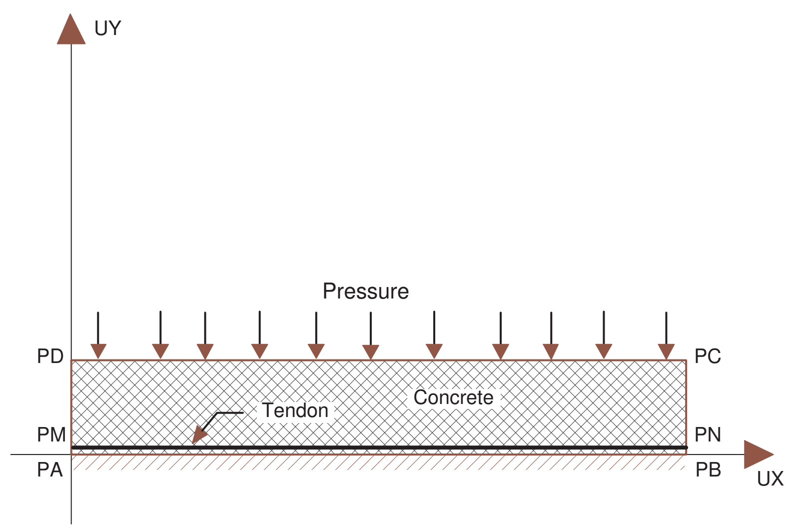
Reference J.S. Fleuret 08/96

Version 97' customer version

Model description

Concrete		
E	=	$0.4\ 10^{11}$
NU	=	0.0

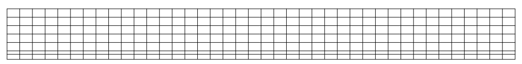
Tendons		
E	=	$2.0\ 10^{11}$
SECT	=	0.02



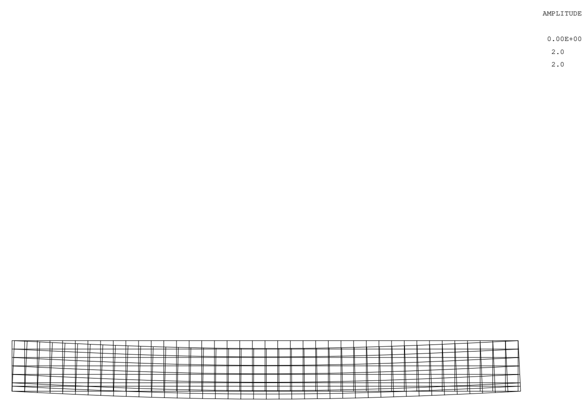
Test precont1

Results

CASTEM FIGURES

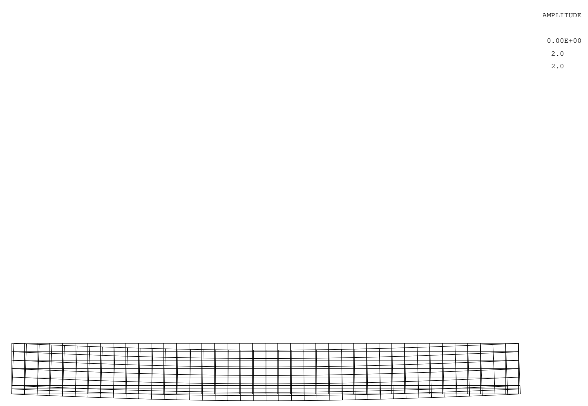


MESHING OF THE BEAM



Deformed shape of concrete under pressure

MESHING OF THE REINFORCEMENT AFTER KABL



Deformed shape of reinforced concrete under pressure

```
*        Test Precont1.dgibi: Jeux de données       *
*        -----------------------------------        *
*                                                   *
*---------------------------------------------------*
* CALCUL AVEC ARMATURE PASSIVE.
* PROGRAMME UTILISANT :
*    - KABL
*    - EQUI RIGI
*
*---------------------------------------------------*
*  J.S. Fleuret 08/96
*---------------------------------------------------*
titr 'precont1.dgibi';
opti echo 1;
graph='n';
opti dime 2 elem qua4 mode plan;
*
* GEOMETRIE
*
*---BETON
pa=0.0 0.0;
pb=10.0 0.0;
pc=10.0 1.0;
pd=0.0 1.0;
nb1=40;
nb2=6;
c1=pa droi nb1 pb;
c2=pb droi nb2 pc;
c3=pc droi nb1 pd;
c4=pd droi nb2 pa;
sur1=dall c1 c2 c3 c4;
*---ARMATURE
pm=0.0 0.1;
pn=10.0 0.1;
***nca = 17 ;
nca = 10 ;
d1 = pm droi nca pn;
*---GEOMETRIE TOTALE
* MODELE
*
modb=modl sur1 mecanique elastique;
d2= (kabl sur1 d1  ) coul roug ;
mod2=modl d2 mecanique elastique armature barr;
*
********d2 = extr mod2 mail;
*
d5= sur1 et d2 ;
si(neg graph 'n');
 titr 'Maillage armature apres KABL';
 trac elem d2;
finsi;
*
* CARACTERISTIQUES
*
mat1=matr modb youn 0.4e11 nu 0.0;
mat2=matr mod2 youn 2.0e11 sect 0.02;
mt=mat1 et mat2;
*
* CONDITIONS AUX LIMITES
*
cl1 = bloq pa ux;
cl2 = bloq pa uy;
cl3 = bloq pb uy;
cltot = cl1 et cl2 et cl3;
*
* CHARGEMENT EN PRESSION
*
cha0 = pres mass modb c3 2.e6;
*
* VERIFICATION QUE PREC ET EQUI FORCES
* NE PLANTE PAS S'IL MANQUE DES VALEURS
* on n'utilise pas le chargement de precontrainte
* dans la suite du programme puisque ce calcul
* concerne une armature passive
*
pre1=prec mod2 mat2 0.4e6 pm;
cha1 = -1 * (bsig pre1 mod2 mat2) ;
list cha1 ;
*
* RESOLUTION BETON SOUS PRESSION
*
*---resolution
ri0 = rigi modb mat1;
rig0cl = ri0 et cltot;
dep0 = reso rig0cl cha0;
*  dep0a = proi arma dep0 modb mod2;
   dep0a = proi  d2  ( chan cham dep0 modb ) ;
*---trace de la deformee
de0 = deform (sur1 et d2) dep0 0.  blan;
de1 = deform sur1 dep0 2.0 roug;
de1a = deform d2 dep0a 2.0 roug;
si(neg graph 'n');
 titr 'deformee du beton sous pression';
 trac (de0 et de1 et de1a);
finsi;
*---affichage des valeurs de fleche
pf = sur1 poin proc (5. 0.);
uf = extr dep0 uy pf;
err0 = abs(((abs(uf) - 0.07794)/0.07794)*100);
mess err0;
*---Code de bon fonctionnement
si ( err0 <  6.E-3  ) ;
   erre 0 ;
sinon ;
   erre 5 ;
finsi ;
*
* RESOLUTION AVEC BETON ET ARMATURE SOUS PRESSION
*
*---resolution
rib = rigi modb mat1;
rik = rigi mod2 mat2 ;
tbzon = extr mod2 ZONE ;
makabl = tbzon . 2 ;
ra = rela makabl ACCROCHER sur1 ;
ritot = rib et rik et ra et cltot ;

fotot = cha0 ;

dep1=reso ritot fotot ;
* dep1a = proi arma dep1 modb mod2;
  dep1a = proi  d2  ( chan cham dep1 modb ) ;
*---trace de la deformee

de0=deform (sur1 et d2) dep1 0.  blan;
de1=deform sur1 dep1 2.0 rouge;
de1a=deform d2 dep1a 2.0 rouge;
si(neg graph 'n');
 titr 'deformee avec beton et armature sous pression';
 trac (de0 et de1 et de1a);
finsi;

*---affichage des valeurs de fleche

pf = sur1 poin proc (5. 0.);
uf = extr dep1 uy pf;
```

```
mess 'Déplacement calculé =' uf ;
err1 = abs(((abs(uf) - 0.06695)/0.06695)*100);
mess 'Erreur relative commise =' err1 '%' ;

*---Code de bon fonctionnement

si ( err1 <  6.E-3  ) ;
   erre 0 ;
sinon ;
   erre 5 ;
finsi ;
*
* FIN DU FICHIER
*
fin;
*-------------------------------------------------------
```

Test precont1 **Comments**

1. The ELASTIC ARMATURE material

```
MOD1 = MODE D1 MECANIQUE ELASTIQUE ARMATURE BARR ;
MAT2 = MATE MOD2 YOUN 2.0E11 SECT 0.02 ;
```

This model concerns the tendons of reinforced concrete (BARR on SEG2). The parameter names for an ELASTIQUE ARMATURE material are as follow:
In the case of passive tendons, the parameters are:
'YOUN' : Young's modulus
'SECT' : section of the tendon
In the case of active tendons (surrounding concrete), the additionnal parameters to be defined are:
For the loss of prestress by friction:
'FF ' : ratio of angular friction (0.18 rd-1)
'PHIF' : ratio of linear friction (0.002 m-1)
For the loss of prestress at the anchoring point:
'GANC' : sliding at the anchoring point (0.0)
For the loss of tensile stress of the tendon:
'RMU0' : ratio of the loss of tensile stress of the tendon (0.43)
'FPRG' : stress of warranted rupture (1700.e6 Pa)
'RH10' : loss of tensile stress after 1000 hours (2.5

2. The KABL operator

```
MODB = MODE SUR1 MECANIQUE ELASTIQUE ;
MOD1 = MODE D1 MECANIQUE ELASTIQUE ARMATURE BARR ;
MOD2 = KABL MODB MOD1 ;
```

The KABL operator calculates the intersection points between a concrete mesh and cables and creates new uniaxial tendon elements accordingly. The parameters to input are:
MODB : model object associated to concrete (MMODEL type)
MOD1 : model object associated to tendons (MMODEL type)
MOD2 : resulting model object (MMODEL type)
The KABL operator is usually used with the EQUI, PREC and PROI operators.

3. The EQUI operator

```
RI1  = EQUI RIGI MODB MOD2 MT ;
CHA1 = EQUI FORCES MODB MOD2 PRE1 ;
```

The operator EQUI calculates the stiffness of tendons equivalent to prestresses expressed at the nodes of the concrete.
'RIGI' : key word to indicate that the calculation of the tendon to the stiffness is required.
'FORCES' : key word to indicate that the calculation of the equivalent forces is required.
MODB : model object associated to concrete (MMODEL type)
MOD2 : model object associated to tendons (MMODEL type)
MT : field of material properties of concrete and tendons (MCHAML type, CARACTERISTIQUES subtype)
RI1 : stiffness matrices (RIGIDITE type , RIGIDITE subtype)
PRE1 : stress field (MCHAML type, CONTRAINTES subtype)
CHA1 : force field (CHPOINT type)

4. The PREC operator

```
PRE1 = PREC MOD2 MAT2 0.4E6 PM ;
```

The PREC operator calculates the loss of prestress and the actions of a tendon on the surrounding concrete.

MOD2	:	model object associated to the tendon (MMODEL type)
MAT2	:	field of caracteristics associated to the tendon (MCHAML type, CARACTERISTIQUES subtype)
0.4E6	:	value of the prestressing force applied to the end of the tendon (FLOTTANT type)
PM	:	anchoring node (POINT type)
PRE1	:	resulting stress field (MCHAML type, CONTRAINTES subtype)

5. The PROI operator

```
DEP1A  = PROI ARMA DEP1 MODB MOD2 ;
```

With the "ARMA" option, the PROI operator projects onto a reinforcement (model MOD2), the displacements of a concrete structure (model MOD1).
The interpolation is done using shape functions of the concrete elements.

DEP1	:	displacements of the concrete structure
MODB	:	model of the concrete structure
MOD2	:	model of the reinforcement

8.3 Test precont2 Description sheet

Test name precont2

Calculation type REINFORCED ELASTIC MECHANICS - BARR - 2D

Finite element type QUA4

Topic Calculation with active reinforcement

The structure is a reinforced concrete beam. The reinforcement is made of tendon. The beam is subjected to pressure on its upper surface, and embedded from its lower side. **Goal** Test the KABL, EQUI RIGI and

PROI ARMA operators with passive reinforcement.

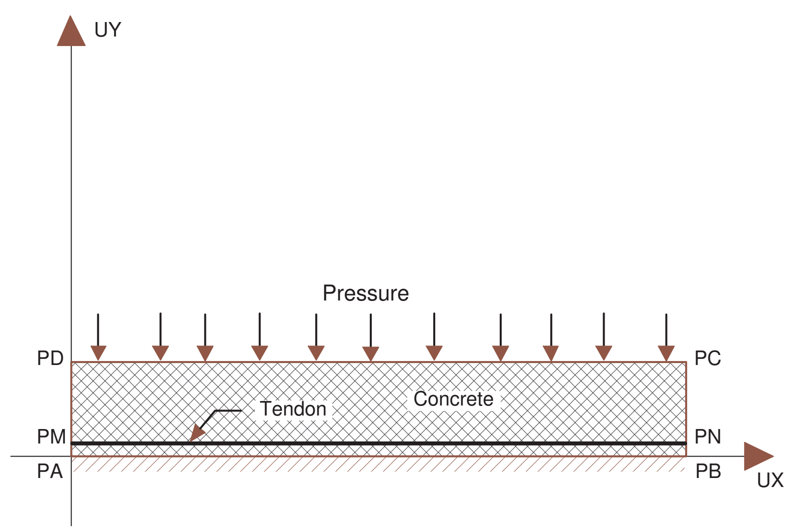
Reference J.S. Fleuret 08/96

Version 97' customer version

Model description

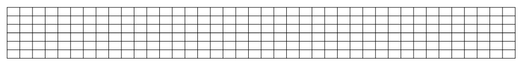
Concrete		
E	=	$0.4\ 10^{11}$
NU	=	0.0

Tendons		
E	=	$2.0\ 10^{11}$
SECT	=	0.02
FF	=	0.18 *or* 0.0
PHIF	=	0.002 *or* 0.0
GANC	=	0.0
RMU0	=	0.43
FPRG	=	$1700.\ 10^{9}$
RH10	=	2.5

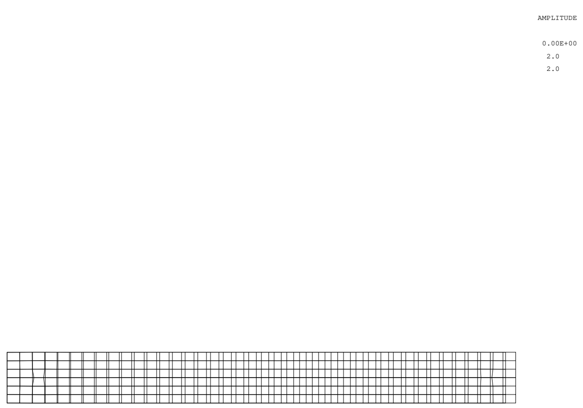


Test precont2 **Results**

CASTEM FIGURES


MESHING OF THE BEAM

MESHING OF THE REINFORCEMENT AFTER KABL



Deformed shape of concrete under prestress

```
*         Test Precont2.dgibi: Jeux de données       *
*         ------------------------------------       *
*                                                    *
*----------------------------------------------------*
* CALCUL AVEC ARMATURE ACTIVE.
* PROGRAMME DE VERIFICATION DE :
*    - PREC
*    - EQUI FORC
*----------------------------------------------------*
*
*----------------------------------------------------*
titr 'precont2.dgibi';
opti echo 1;
graph='n'    ;
opti dime 2 elem qua4 mode plan;
*
* GEOMETRIE
*
long = 10.0;
*---BETON
h = 1. ;
h1= .9 ;
*h1 = 1. -  (1./6.1);
pa=0.0 0.0;
pb=long 0.0;
pc=long h.;
pd=0.0 h;pn = 0. (h - h1)  ;
nb1=40;
nb2=6;


c1=pa droi nb1 pb;
c2=pb droi nb2 pc;
c3=pc droi nb1 pd;
c4=pd droi nb2 pa;
sur1=dall c1 c2 c3 c4;
*---ARMATURE
pm = long (h - h1)  ;
pn = 0.   (h - h1)  ;
*nca = 20 ;
 nca = 17 ;
d1 = pm droi nca pn;
* d1 = pn droi nca pm;
*---GEOMETRIE TOTALE

si(neg graph 'n');
 titr 'massif' ( sur1 nbel ) 'cable initial' (d1 nbel);
 trac elem ( sur1 et d1) ;
finsi;
*
* MODELE
*
modb=modl sur1 mecanique elastique;
d2 = (kabl sur1 d1 ) coul roug ;
mod1 = modl d1 mecanique elastique armature barr;
mod2 = modl d2 mecanique elastique armature barr;
d5= sur1 et d2 ;
si(neg graph 'n');
 titr 'Maillage armature apres KABL' (d2 nbel) ;
 trac d2 elem   ;

finsi;
*
* CARACTERISTIQUES DU BETON
*
Eb = 0.4e11 ;
matb=matr modb youn Eb nu 0.0;
Ea = 2.0e11 ;ssect = .02 ;
```

```
*
*   TENSION DU CABLE
SIG0 = 1200.E6 ; tens = sig0 * ssect ;

mat2= (mate mod2 youn Ea ) et ( cara mod2 sect ssect );

*     Pour la methode accro on definit               *
*     un materiau sur le renfort initial             *

mat1= (mate mod1 youn Ea ) et ( cara mod1 sect ssect );
*
* CONDITIONS AUX LIMITES
*
cl1 = bloq pa ux;
cl2 = bloq pa uy;
cl3 = bloq pb uy;
cl4 = rela ense ux c2;
cl5 = rela ense ux c4;
cltot = cl1 et cl2 et cl3 ;
*



*
* CHARGEMENT
TA = TABLE ;
TA . FPRG = 0. ;
pre1=(prec mod2 mat2 tens TA  pm)  ;
si ( neg graph 'n') ;
titre 'Precontaintes sans pertes  (mprecont20.dgibi) ';
ev1 = evol roug chpo ( chan chpo pre1 mod2) EFFX
      ( inve d2) ;
dess ev1 ;
finsi ;

 chac = -1 * (bsig pre1 mod2 mat2) ;
 cha1 = -1 * (equiv forces modb mod2 pre1 );

 rchac = resu chac ;
 rcha1 = resu cha1 ;
list rchac ;
list rcha1 ;
faccro = (redu chac pm) resu abs maxi ;
faequi = (redu cha1 c2) resu abs maxi ;

err1 = abs(((faequi - tens )/tens)*100) ;
err2 = abs(((faccro - tens )/tens)*100) ;

*---Code de bon fonctionnement
si ((err1 < 1.) et ( err2 < 1.))  ;
   erre 0 ;
sinon ;
   erre 5 ;
finsi ;



si(neg graph 'n');

uu = maxi ( abs chac) avec ( mots FX) ;
ampv = 2./uu ;
TITRE ' resultante chargement bsigma '  faccro ;
vvbsi = vecteur chac FX FY  ampv   vert ;
trac sur1 vvbsi ;
vvv   = vecteur cha1 FX FY  ampv  vert ;
TITRE ' resultante chargement equiv prec ' faequi  ;
trac sur1 vvv ;
finsi;
```

```
*
* RESOLUTION BETON SOUS PRECONTRAINTE
*
*---resolution
 rib = rigi modb matb;
     mt = matb et mat2    ;
     raeq = equi rigi modb mod2 mt ;

     depequi = reso (rib  et cltot ) cha1;
*  rik est la raideur  des cables  pour modele accro
     rik = rigi mod1 mat1 ;
     racc = rela d2 ACCROCHER sur1 ;
     depacc = reso (rib  et racc  et cltot ) chac;

*----------trace de la deformee
uma1 = maxi depequi avec ( mots UY) ;
uma2 = maxi depacc avec  ( mots UY) ;


*---Code de bon fonctionnement

err10 = abs(((uma1 - uma2)/uma1)*100) ;

si  ( err10 < 1. ) ;
   erre 0 ;
sinon ;
   erre 5 ;
finsi ;



amp1 = .2/uma1 ;
de0=deform (sur1 et d2) depequi 0.  blan;

dep0a = proi d2 (chan  cham  depequi  modb NOEUDS) ;
dep0b = proi d2 (chan  cham  depacc   modb NOEUDS) ;

de1=deform sur1 depequi amp1 bleu ;

de2 = deform sur1 depacc amp1 bleu ;

de1a=deform d2 dep0a amp1 vert ;
de1b=deform d2 dep0b amp1 turq ;

*   CACLUL DES  PRECONTRAINTES BETON ' ;

si(neg graph 'n');

 titr ' precontrainte equi uy max = ' uma1
       ' tension cable '  faequi ;
 trac (de0 et de1 et de1a);
 titr ' precontrainte accr uy max = ' uma2
       ' tension cable '  faccro ;
 trac ( de0 et de2 et de1b) ;

finsi;


***  AJOUT  D'UNE CHARGE SUR LA FACE SUPERIEURE    ***
 b= 1.;
 X = ((h*h*Eb )+ (2*h1*Ea*ssect))/
     ((2*h*Eb)+(2*Ea*ssect)) ;
 ii = (((x**3) + ((h-x)**3))/3 )+((Ea*ssect*(h1-x)*
      (h1-x))/Eb);
 prsup = press mass modb c3 2.e6;
 FTS = 2.E6 * long ;

*----------- FLECHE ANALYTIQUE SANS PRECONTRAINTE
  dmax = 5.*FTS*(long**3)/(384*Eb*ii) ;
```

```
  mess ' fleche analytique  ' dmax ;

* ----------  CALCUL PAR EQUI RIGI   ET    RELA ACCRO

dep2equi = reso ( rib et  raeq  et cltot       ) prsup;
dep2accr = reso ( rib et  racc  et rik et cltot) prsup;
uyma1 =  mini dep2equi  avec ( mots uy) ;
uyma2 =  mini dep2accr avec ( mots uy) ;
amp = .2/(abs uyma1) ;

*---Code de bon fonctionnement
err1 = abs(((uyma1 + dmax )/dmax)*100) ;
err2 = abs(((uyma2 + dmax )/dmax)*100) ;

si ( ( err1 <  1.  ) et ( err2  <  1.   ) ) ;
   erre 0 ;
sinon ;
   erre 5 ;
finsi ;


defequi=deform sur1 dep2equi amp rouge;
defaccr=deform sur1 dep2accr amp rouge;

si(neg graph 'n');
 titr ' force repartie / equiv uy max = ' uyma1
       ' analytique ' dmax ;
 trac (de0 et defequi );
 titr ' force repartie / accro uy max = ' uyma2
       ' analytique ' dmax ;
 trac (de0 et defaccr );
finsi;


* MEME PRECONTRAINTE AVEC PERTE DE
*    PRECONTRAINTE DUE AU FROTTEMENT
*  S = So * (1 - exp(-ff*alpha-phif*x))
*
*--- avec ff = 0.18 et phif = 0.002
    mat2=(mate mod2 youn 2.0e11 ff 0.18 phif 0.2
         ganc 0.00 rmu0 0.43 fprg 1700.e9 rh10 2.5 )
         et ( cara mod2 sect .02 ) ;


pre2= (prec mod2 mat2 tens pm) ;
cha2 = -1 * (equiv forces modb mod2 pre2 );
chac = -1 * (bsig pre2 mod2 mat2) ;
uu = maxi ( abs cha2) avec ( mots FX) ;
ampv = 2./uu ;

si ( neg graph 'n' ) ;
titre 'Precontaintes avec pertes  (mprecont20.dgibi) ';
ev2 = evol roug chpo ( chan chpo pre2 mod2) EFFX
      ( inve d2) ;
dess ev2 ;
vv2   = vecteur cha2 FX FY  ampv  vert ;
TITRE '  chargement avec pertes de precontrainte '   ;
trac sur1 vv2 ;

finsi ;
*------------    DEFORMATIONS INITIALES ---------------


  depert1  = reso (rib  et cltot ) cha2;
  depert2  = reso (rib  et racc et cltot ) chac ;

  de1p=deform sur1 depert1 amp1 bleu ;
  de2p=deform sur1 depert2 amp1 vert ;
```

```
  uma3 = maxi depert1 avec ( mots UY) ;
  uma4 = maxi depert2 avec ( mots UY) ;
si ( neg graph 'n') ;
 titre ' def init. Avec pertes de precontrainte equi '
       uma3 ;
 trac ( de0 et  de1p) ;
 titre ' def init. Avec pertes de precontrainte accro '
        uma4 ;
 trac ( de0 et  de2p) ;
finsi ;

*---------- DEFORMATIONS TOTALES   PETITS DEPLACEMENTS

 dept1 = depert1 + dep2equi ;
 dept2 = depert2 + dep2accr ;
  uma5 = mini dept1 avec ( mots UY) ;
  uma6 = mini dept2 avec ( mots UY) ;

si ( neg graph 'n') ;
 titre 'def totale Avec pertes de precontrainte equi '
       uma5 ;
 trac ( de0 et  de1p) ;
 titre 'def totale Avec pertes de precontrainte accro '
       uma6 ;
 trac ( de0 et  de2p) ;
finsi ;


 fin ;
```

Test precont2 **Comments**

- The RELA operator

```
CL4  = RELA ENSE UX C2 ;
```

The ENSE option of the RELA operator enables the user to construct the stiffness CL4 (RIGIDITE type) associated with a global motion : the value on the degree of freedom UX (MOT type) is imposed as being identical for all the nodes of C2 (MAILLAGE type).

8.4 Test precont3 Description sheet

Test name precont3

Calculation type REINFORCED ELASTIC MECHANICS - BARR - 2D

Finite element type QUA4

Topic Calculation with active reinforcement and different loss of prestress

The structure is a reinforced concrete beam. The reinforcement is made of tendon. The beam is subjected to pressure on its upper surface, and embedded from its lower side. **Goal** Test the PREC operators with passive

reinforcement.

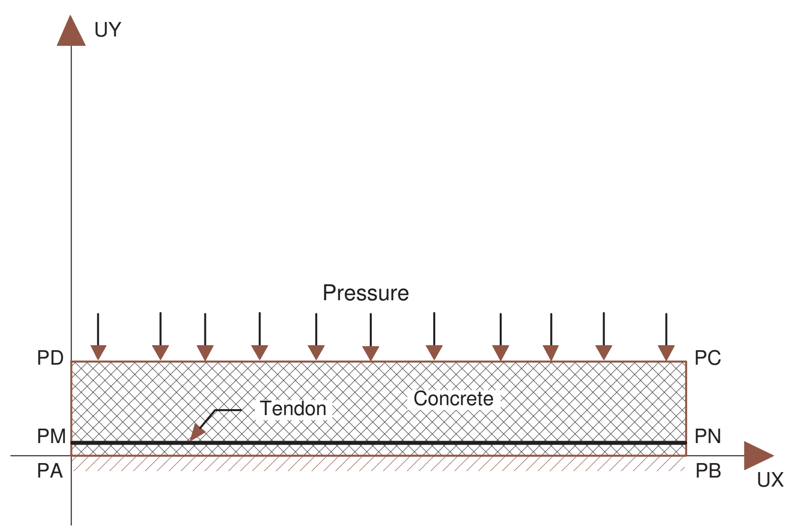
Reference J.S. Fleuret 07/96

Version 97' customer version

Model description

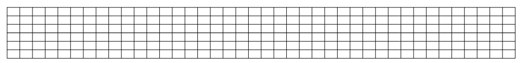
Concrete		
E	=	$0.4\ 10^{11}$
NU	=	0.0

Tendons	without loss	without loss	with loss for friction	with loss for friction and slipping	with loss for friction slipping and relaxation
E	= $2.1\ 10^{11}$	= $2.1\ 10^{11}$	= $2.1\ 10^{11}$	= $2.1\ 10^{11}$	= $2.1\ 10^{11}$
SECT	= 0.02	= 0.02	= 0.02	= 0.02	= 0.02
FF	= 0.0	= 0.0	= 0.18	= 0.18	= 0.18
PHIF	= 0.0	= 0.0	= 0.002	= 0.002	= 0.002
GANC	= 0.0	= 0.0	= 0.0	= 0.005	= 0.005
RMU0	= 0.43	= 0.43	= 0.43	= 0.43	= 0.43
FPRG	= $1700.\ 10^{9}$	= $1700.\ 10^{9}$	= $1700.\ 10^{9}$	= $1700.\ 10^{9}$	= $1700.\ 10^{9}$
RH10	= 2.5	= 2.5	= 2.5	= 2.5	= 2.5



Test precont3 **Results**

CASTEM FIGURES



MESHING OF THE BEAM

MESHING OF THE REINFORCEMENT AFTER KABL

```
*        Test Precont3.dgibi: Jeux de données       *
*        -----------------------------------        *
*                                                   *
*---------------------------------------------------*
* CALCUL AVEC DIFFERENTES PERTES DE PRECONTRAINTE
* PROGRAMME DE VERIFICATION DE :
*    - PREC
*---------------------------------------------------*
*  J.S. Fleuret 07/96  modifie JMB 11/97
*---------------------------------------------------*
titr 'precont3.dgibi';
opti echo 1;
graph='N';
tabu= table ;
tabu.1 = 'MARQ CROI  TITR Sans_pertes' ;
*tabu.2 = 'MARQ TRIA  TITR Sans_pertes_+pre_init' ;
tabu.2 = 'MARQ CARR  TITR Pertes_frott' ;
tabu.3 = 'MARQ LOSA  TITR Pertes_frott_glis' ;
tabu.4 = 'MARQ ETOI  TITR Pertes_frott_glis_rel' ;

opti dime 2 elem qua4 mode plan;
*
* GEOMETRIE
*
long = 10.0;
*---BETON
h = 1. ;h1 = .5 ;
pa=0.0 0.0;
pb=long 0.0;
pc=long h;
pd=0.0  h;
nb1=40;
nb2=4;
c1=pa droi nb1 pb;
c2=pb droi nb2 pc;
c3=pc droi nb1 pd;
c4=pd droi nb2 pa;
sur1=dall c1 c2 c3 c4;
*---ARMATURE
pm=long ( h - h1) ;
pn= 0.0 ( h - h1) ;
nca =15 ;d1 = pm droi nca pn;

d2 = pm droi nca pn;
*---GEOMETRIE TOTALE

si(neg graph 'N');
 titr 'Maillage de la poutre';
 trac elem (sur1 et d1);
finsi;
*
* MODELE
*
modb=modl sur1 mecanique elastique;
d2 = (kabl sur1 d1  ) coul roug ;
mod2 =modl d2 mecanique elastique armature barr;

si(neg graph 'N');
 titr 'Maillage armature apres KABL' ( d2 nbel);
 trac elem d2;
finsi;
pmil=  (d2 point proche (5. 0.)) ;
x = NORM  ( PM moins pmil) ;
*
* CARACTERISTIQUES DU BETON
*
mat1=matr modb youn 0.4e11 nu 0.0;
*
```

```
* * CONDITIONS AUX LIMITES
* *
* cl1 = bloq pa ux;
  cl2 = bloq pa uy;
  cl3 = bloq pb uy;
  cl4 = rela ense ux c2;
  cl5 = rela ense ux c4;
  cltot = cl1 et cl2 et cl3 et cl4 et cl5;
  *      Contrainte de depart du cable  ( pas tension )

  ssect = .02 ;Ea = 2.1e11 ;
  sig0 = 1200.E6 ;tens = sig0 * ssect   ;


  si ( neg graph 'N' ) ;
  MESS 'POUTRE BETON   LONGUEUR 10M  HAUTEUR 1M  ' ;
  MESS 'ARMATURE   SECTION                         ' SSECT;
  MESS 'TENSION SOUS VERIN   COTE GAUCHE         ' TENS ;
  sauter 3 lignes ;
  mess '********************************************* ';
  mess 'PRECONTRAINTE   SANS PERTE   ' ;
  mess '********************************************* ';
  finsi ;
  *mat2=(mate mod2 youn Ea  ff 0.0 phif 0.0
  *     ganc 0.0 rmu0 0.43 fprg 1700.e9 rh10 2.5)
  *     et ( cara mod2 sect ssect )  ;
  mat2=(mate mod2 youn 2.0e11 ) et
                      ( cara mod2 sect .02 ) ;
  tabi = table ;
  tabi.'FF' =  0.0 ;
  tabi.'PHIF' =  0.0 ;
  tabi.'GANC' = 0.0 ;
  tabi.'RMU0' = 0.0 ;
  tabi.'FPRG'= 0.0 ;
  tabi.'RH10'= 0.0 ;

  pre1=prec mod2 mat2 tabi tens ;
  cpre1 = chan chpo  pre1 mod2 ;
  prec100 = extr cpre1 effx PN;
  prec050 = extr cpre1 effx pmil;
  si ( neg graph 'N') ;
  titre ' precontrainte sans perte 1 ' ;
  ev1 = evol roug  chpo cpre1 EFFX d2 ;
  dess ev1 xbor 0. 10. ;
  finsi ;

  err0 = ( (abs(prec100-tens)*100/tens)
         + (abs(prec050-tens)*100/tens) )/2;
  mess err0 '%';
  *---Code de bon fonctionnement
  si ( err0 <  0.005  ) ;
     erre 0 ;
  sinon ;
     erre 5 ;
  finsi ;
  *
  si ( neg graph 'N' ) ;
  mess '********************************************* ';
  mess ' PRECONTRAINTE   avec  PERTE   PAR FROTTEMENT ' ;
  mess '********************************************* ';
  finsi ;

  *
  af = 0.18 ;
  aphif = 0.02 ;
  ag = 0. ;
  amu = 0. ;
  afr = 0. ;
```

```
arh = 0.;
t10 = tens * (exp ( aphif * 10. * -1.)) ;
t5  = tens * (exp ( aphif *  5. * -1.)) ;
*
*mat2=(mate mod2 youn Ea ff af  phif aphif
*     ganc ag rmu0 amu  fprg afr rh10 arh ) et
*     ( cara mod2 sect ssect ) ;
tab1 = table ;
tab1.'FF' =  af  ;
tab1.'PHIF' =  aphif;
tab1.'GANC' = 0.0 ;
tab1.'RMU0' = 0. ;
tab1.'FPRG'=0. ;
tab1.'RH10'= 0. ;
pre1=prec mod2 mat2  tab1 tens  ;
cpre1 = chan chpo  pre1 mod2 ;
prec101 = extr cpre1 effx PN;
prec051 = extr cpre1 effx  pmil;
si ( neg graph 'N' ) ;
mess '10 m  ' prec101 'T analytique ' t10
     ' SIG ' (t10/ssect);
mess ' 5 m  ' prec051 'T analytique ' t5
     ' SIG ' (t5/ssect);
titre ' precontrainte avec frottement ' ;
ev3 = evol vert  chpo cpre1 EFFX d2 ;
dess ev3  xbor 0. 10.;
finsi ;

err1 = ( (abs(prec101-t10)*100/t10)
      + (abs(prec051-t5)*100/t5) )/2;
mess err1 '%';


*---Code de bon fonctionnement
si ( err1 <  1.  ) ;
   erre 0 ;
sinon ;
   erre 5 ;
finsi ;

si ( neg graph 'N' ) ;
mess '********************************************** ';
mess ' PRECONTRAINTE   avec  PERTE   PAR FROTTEMENT ' ;
mess '                               par GLISSEMENT ' ;
mess '********************************************** ';
finsi ;
*
* CALCUL DE LA PRECONTRAINTE
* AVEC PERTE DUE : AU FROTTEMENT
*                  AU GLISSEMENT
*
ag = 0.005 ;
tt10 = t10 ;tt5= t5 ;
alamb = (ag * Ea/(sig0  * aphif)) ** .5  ;
si ( 10. < alamb ) ;
dt10 = 2. * tens * aphif * (alamb - 10.) ;
tt10 = tt10 - dt10 ;
finsi ;
si ( 5. < alamb) ;
dt5  = 2. * tens * aphif * (alamb -  5.)  ;
tt5  = t5  - dt5  ;
finsi ;



*mat2=(mate mod2 youn Ea ff af  phif aphif
*     ganc ag rmu0 amu  fprg afr rh10 arh ) et
*     ( cara mod2 sect ssect ) ;
tab2 = table ;
tab2.'FF' =  af  ;
tab2.'PHIF' =  aphif;
tab2.'GANC' = ag ;
tab2.'RMU0' = amu ;
tab2.'FPRG'= afr ;
tab2.'RH10'= arh  ;
pre1= prec mod2 mat2   tab2  tens  ;
cpre1 =  chan chpo pre1 mod2 ;
prec102 = extr cpre1 effx PN;
prec052 = extr cpre1 effx pmil;

si ( neg graph 'N') ;
mess ' 10 metres lamb ' alamb ' tens ' tt10 ;
mess '  5 metres lamb ' alamb ' tens ' tt5  ;
mess '10 m ' prec102 'T analytique ' tt10
     ' SIG ' (tt10/ssect);
mess ' 5 m ' prec052 'T analytique ' tt5
     ' SIG ' (tt5/ssect);
titre ' precontrainte avec frottement et glissement ' ;
ev4 = evol vert  chpo cpre1 EFFX d2 ;
dess ev4  xbor 0. 10.;
finsi ;

err2 = ( (abs(prec102-tt10)*100/tt10)
       + (abs(prec052-tt5)*100/tt5) )/2;
mess err2 '%';
*---Code de bon fonctionnement
si ( err2 <  1.  ) ;
   erre 0 ;
sinon ;
   erre 5 ;
finsi ;

 si ( ega ag 0.) ;
 tt10= t10 ;
 tt5=t5 ;
 finsi ;
si ( neg graph 'N' ) ;
mess '********************************************** ';
mess ' PRECONTRAINTE   avec  PERTE   PAR FROTTEMENT  ';
mess '                               par GLISSEMENT  ';
mess '                               par RELAXATION  ';
mess '********************************************** ';
finsi ;
amu = .43 ;
afr = 1700.e6 ;
arh = 2.5 ;

ttt10 = tt10 ;ttt5=tt5 ;
p10 = 0.06 * (((tt10/(afr*ssect)))-amu) * tt10 * arh ;
p5 =  0.06 * (((tt5/(afr*ssect)-amu))) * tt5 * arh ;
si ( p10 > 0. ) ;
ttt10 = ttt10 - p10 ;
finsi ;
si ( p10 > 0. ) ;
ttt5 = ttt5 - p5 ;
finsi ;
tab3 = table ;
tab3.'FF' =  af  ;
tab3.'PHIF' =  aphif;
tab3.'GANC' = ag ;
tab3.'RMU0' = amu ;
tab3.'FPRG'= afr ;
tab3.'RH10'= arh  ;
*mat2=(mate mod2 youn Ea  ff af phif aphif
*     ganc ag rmu0 amu fprg afr rh10 arh) et
*     ( cara mod2 sect ssect ) ;
```

```
pre1=prec mod2 mat2 tens tab3;
cpre1 = chan chpo  pre1 mod2 ;
prec103 =  extr cpre1 effx PN;
prec053 =  extr cpre1 effx pmil;
si ( neg graph 'N') ;
mess '10 m  ' prec103 'T analytique ' ttt10
     ' SIG ' (ttt10/ssect) ;
mess ' 5 m  ' prec053 'T analytique ' ttt5
     ' SIG ' (ttt5/ssect);
titre 'precontrainte avec frottement
glissement et relaxation';
ev5 = evol vert  chpo ( chan chpo  pre1 mod2) EFFX d2 ;
dess ev5 xbor 0. 10. ;
finsi ;

err3 = ( (abs(prec103-ttt10)*100/ttt10)
       + (abs(prec053-ttt5)*100/ttt5) )/2;
mess  err3  ' en %';
*---Code de bon fonctionnement
si ( err3 <  4.  ) ;
   erre 0 ;
sinon ;
   erre 5 ;
finsi ;
*
* FIN DU FICHIER
*
 si ( neg graph 'N') ;
 dess ( ev1  et ev3 et ev4 et ev5) xbor 0. 10.
      lege tabu ;
 finsi ;

 fin ;
```

Chapter 9

POROSITY

9.1 Test pore1 Description sheet

Test name pore1

Calculation type ELASTOPLASTIC MECHANICS - DRUCKER-PRAGER MODEL IN POROSITY - MODE AXIS **Finite element type** QUAP

Topic Porous thick cylinder

The structure is an elastoplastic thick cylinder. It follows a Drucker-Prager law with non-associated yield. We release the stress within the hole.

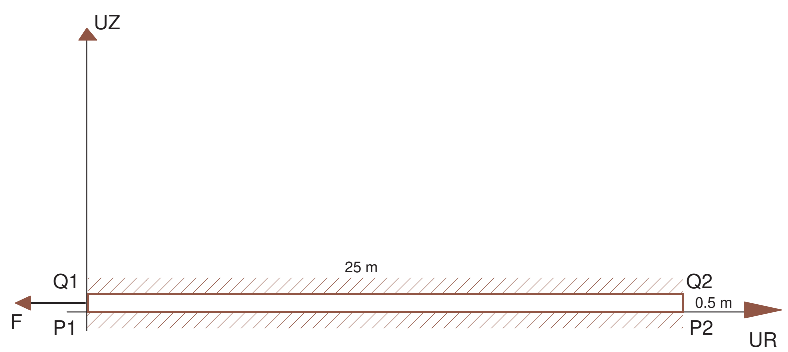
Goal Calculate the convergence of the hall and the stress along the circumference

Reference Benchmark INTERCLAY 1.1 Variante 2

Version 97' customer version

Model description

E	=	200.13E06	Young's modulus
NU	=	0.25	Poisson's ratio
RHO	=	0.	density
ALPH	=	0.	thermal expansion coefficient
MOB	=	7500E6	Biot's modulus
COB	=	1.	Biot's coefficient
PERM	=	4.E-12	intrinsic permeability
VISC	=	4.	fluid dynamic viscosity
ALPM	=	0.	coefficient of the pressure-temperature coupling



Test pore1 **Results**

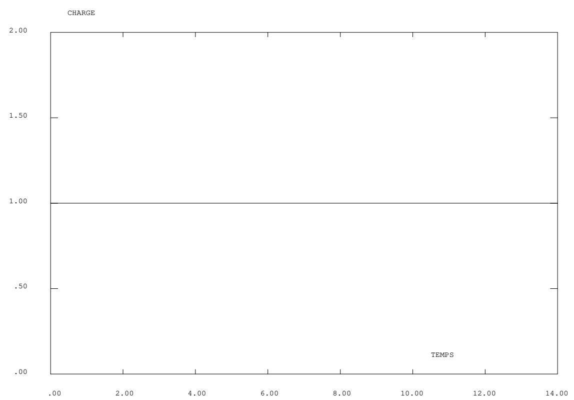
RESULTS

	Hall convergence	Stress along the circumference
Reference (*)	-0.266 m	-3.47E6 Pa
Castem	-0.26586 m	-3.46667E6 PA
Error	5.21515E-4 %	9.59243E-4 %

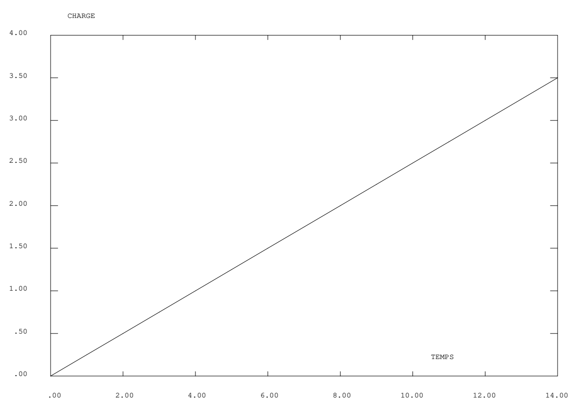
(*) The reference values are calculated with castem2000 which agree with INTERCLAY's results.

CASTEM FIGURES

A THICK CYLINDER IN POROUS MIDDLE



INITIAL STRESSES



PRESSURE

```
*           Test Pore1.dgibi: Jeux de données        *
*           --------------------------------         *
*                                                    *
SAUT PAGE;
*
******************************************************
*                                                    *
*  TEST PORE1                                        *
*                                                    *
*  CYLINDRE EPAIS EN MILIEU POREUX                   *
*                                                    *
*  REFERENCE : Benchmark INTERCLAY 1.1  Variante 2   *
*                                                    *
*     Le milieu est elastoplastique, modele          *
*     Drucker-Prager non associe                     *
*     On decharge la contrainte a l'interieur du trou *
*                                                    *
*     On se propose de caculer la convergence du trou *
*     valeur de reference  : -0.266 m                *
*     ainsi que la contrainte circonferentielle au   *
*     bord du trou                                   *
*     valeur de reference  : -3.47E6 Pa              *
*     Il s'agit de valeurs calculees par castem2000  *
*     et en accord avec les resultats de INTERCLAY   *
*                                                    *
******************************************************
*
OPTI ECHO 0;
TITRE   ' TEST PORE1  ' ;
GRAPH = 'N';
OPTIO DIME 2 MODEL AXIS   ELEM QUA8  ;
*
*====================================================*
*             geometrie : maillage                   *
*====================================================*
*
* Dimension en metres
*
DENS 0.5;
P1 = 2.5 0. ; P2 = 25. 0. ; Q1 = 2.5 0.5 ;
Q2 = 25. 0.5; PP1 = 5 0.  ; QQ1 = 5 0.5 ;
LI1 = P1 D  P2 'DINI' 0.25 'DFIN' 2.5 ;
LI2 =  P2 D Q2 ;
LI3  = INVE ( Q1 D Q2 'DINI' 0.25 'DFIN' 2.5 );
LI4  = Q1 D P1 ;
LI  = LI1  ET LI2  ET LI3  ET LI4  ;
CADRE = DALLER LI1  LI2  LI3  LI4  PLAN ;
SI (NEG GRAPH 'N');
   TRAC CADRE ;
FINSI;
*
*====================================================*
*      modele - materiau - caracteristiques en Pa    *
*                conditions aux limites              *
*====================================================*
*
 MO=MODL CADRE POREUX ELASTIQUE PLASTIQUE
                     DRUCKER_PRAGER ;
*
MAT=MATR MO YOUNG 200.13E6 NU 0.25 MOB  7500E6 COB 1.
 PERM 4.E-12   VISCO 4.
ALPH 0. KF 0. RHOF 0. ALPF 0. ALPM 0. RHO 0.
 ALFA 0.1270509  BETA 0.57735027 K 0.
 GAMM 0.         DELT 0.57735027
 ETA  0.1270509  MU   0.57735027 KL 0. H 0. ;
*
*  on bloque les faces superieure et inferieure
*
BL1=BLOQ UZ  LI1 ;
BL2=BLOQ UZ  LI3 ;
CL = BL1 ET BL2  ;
*
*====================================================*
*                  Chargement                        *
*====================================================*
*
*   contraintes initiales
*
ZOZO = MANU CHML MO    SMRZ 0.
   SMRR -5.E6 SMTT -5.E6 SMZZ -5.E6 PRES 2.5E6
   TYPE CONTRAINTES  STRESSES ;
FF = BSIGMA MO ZOZO;
EVT1= EVOL MANU 'TEMPS'  (PROG 0   14 )
                'CHARGE' (PROG 1   1  );
CHA1 = CHAR MECA FF EVT1 ;
*
*    pression
*
FPI2 = PRESS MASS MO -1.E6   LI4 ;
EVT2= EVOL MANU 'TEMPS' (PROG 0 PAS 1 14 )
                'CHARGE' (PROG 0 PAS 0.25 3.5) ;
CHA2 = CHAR MECA FPI2 EVT2 ;
CHATT = CHA1 ET CHA2 ;
*
*====================================================*
*   Initialisation de la table pour appel a PASAPAS  *
*====================================================*
*
TAB1 = TABLE;
TAB1 . 'CONTRAINTES' = TABLE;
TAB1 . 'DEPLACEMENTS' = TABLE;
TAB1.'CONTRAINTES' . 0 = ZOZO;
TAB1.'DEPLACEMENTS' . 0 = MANU CHPO CADRE  1 P 2.5E6 ;
LREE = PROG 0 PAS 1 14;
TAB1.'CARACTERISTIQUES' = MAT;
TAB1.'CHARGEMENT' = CHATT;
TAB1.'MODELE' = MO;
TAB1.'TEMPS_CALCULES' = LREE;
TAB1.'BLOCAGES_MECANIQUES' = CL;
*
PASAPAS TAB1;
*====================================================*
*               affichage des resultats              *
*====================================================*
*
MESS ' RESULTATS TEST PORE1 ';
MESS ' -------------------- ';
SAUT 2 LIGN;
MESS
'  On cherche la convergence du trou et la contrainte';
MESS
'  circonferentielle ';
MESS
'  Elles sont comparees a des valeurs de reference ';
SAUT 1 LIGN;
SAUT 1 LIGN;
CONVTH = -0.266 ;
SIGTTH = -3.47E6 ;
LEDEP = PECHE TAB1 DEPLACEMENTS 14 ;
SIGAA = PECHE TAB1 CONTRAINTES  14 ;
SIGAB = CHAN 'CHPO' MO SIGAA ;
CONVXX = EXTR LEDEP UR P1 ;
SIGTXX = EXTR SIGAB SMTT P1 ;
ERGXXA = ( CONVXX - CONVTH ) / CONVTH ;
ERGXXA = ABS ERGXXA ;
ERGXXB = ( SIGTXX - SIGTTH ) / SIGTTH ;
```

```
ERGXXB = ABS ERGXXB ;
*
MESS ' Convergence de reference  :' CONVTH 'm';
MESS ' Convergence calculee      :' CONVXX 'm';
MESS '    Soit un ecart de :' ERGXXA '%';
SAUT 1 LIGN;
MESS ' Contrainte circonferentielle de reference  :'
     SIGTTH 'Pa';
MESS ' Contrainte circonferentielle calculee      :'
     SIGTXX 'Pa';
MESS '    Soit un ecart de :' ERGXXB '%';
*
*
*=====================================================*
*                  code fonctionnement                *
*=====================================================*
*
*    L'ecart maximum entre valeur theorique et calculee
* doit etre inferieure a 5%.
*
SAUT 2 LIGNE;
SI (ERGXXA <EG 5) ;
  SI (ERGXXB <EG 5) ;
    ERRE 0;
  SINON;
    ERRE 5;
  FINSI;
SINON;
  ERRE 5;
FINSI;
*
SAUT 1 LIGN;
*
FIN;
```

Test pore1 **Comments**

1. Parameter name for an ELASTIQUE ISOTROPE material in the case of a POREUX formulation
   The parameters which should be input in the POREUX formulation are as follow:

E	Young's modulus
NU	Poisson's ratio
RHO	density
ALPH	thermal expansion coefficient
MOB	Biot's modulus
COB	Biot's coefficient
PERM	intrinsic permeability
VISC	fluid dynamic viscosity
ALPM	coefficient of the pressure-temperature coupling

2. The Drucker-Prager's model with non-associated yield and stress hardening

```
MO=MODE CADRE POREUX ELASTIQUE PLASTIQUE
                     DRUCKER_PRAGER ;
MAT=MATE MO YOUNG 200.13E6 NU 0.25
  MOB  7500E6     COB 1.
  PERM 4.E-12     VISCO 4.
  ALPH 0. KF 0.   RHOF 0. ALPF 0.  ALPM 0. RHO 0.
  ALFA 0.1270509  BETA 0.57735027  K 0.
  GAMM 0.         DELT 0.57735027
  ETA  0.1270509  MU   0.57735027  KL 0.   H 0. ;
```

The equations of the plastic DRUCKER-PRAGER model are as follow:
–> Notations: S stress tensor
Seq Von Mises equivalent stress
p cumulated equivalent plastic strain
–> initial criterion: ALFA * Tr(S) + BETA * Seq = K
–> ultimate criterion: ETA * Tr(S) + MU * Seq = KL
–> Hardening: dK = H * dp (H in algebraic value)
–> Flow potential: GAMM * Tr(S) + DELT * Seq
The parameters to be defined are: ALFA, BETA, K, ETA, MU, KL, H, GAMM, DE

3. Application of the initial stresses

```
ZOZO  = MANU CHML MO    SMRZ 0.
        SMRR -5.E6 SMTT -5.E6 SMZZ -5.E6 P 2.5E6
        TYPE CONTRAINTES  STRESSES ;
FF    = BSIGMA MO ZOZO;
EVT1  = EVOL MANU TEMPS  (PROG 0   14 )
                  CHARGE (PROG 1   1  ) ;
CHA1  = CHAR MECA FF EVT1 ;
```

The MANU CHML operator creates a field by element (MCHAML type) connected with the MO object (MMODEL type). The key word 'stresses' means that the field by element will be expressed with respect to element stress calculation points.
The BSIGMA operator calculates the nodal force field resulting from the stress field integration.

9.2 Test pore2 Description sheet

Test name pore2

Calculation type ELASTOPLASTIC MECHANICS - DRUCKER-PRAGER MODEL IN POROSITY - MODE AXIS **Finite element type** QUAP

Topic Porous thick cylinder with thermal effects

The structure is an elastoplastic thick cylinder. It follows the Von-Mises model with associated yield. We release the stress within the hole, and we set a temperature yield.

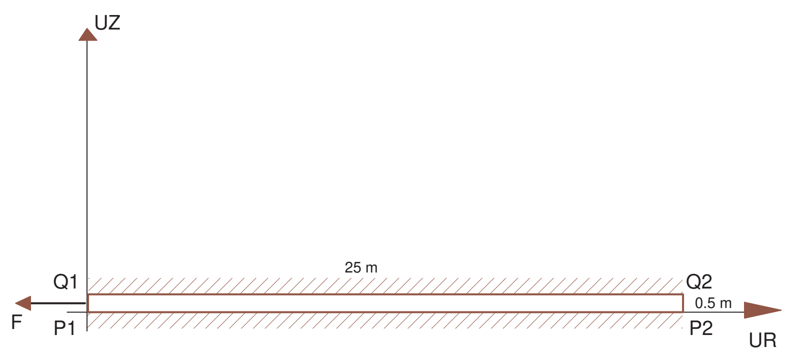
Goal Calculate the convergence of the hole and the stress along the circumference

Reference Benchmark INTERCLAY 1.4 Basic Model

Version 97' customer version

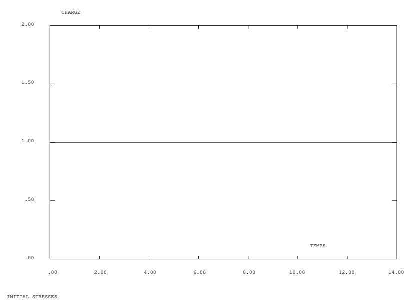
Model description

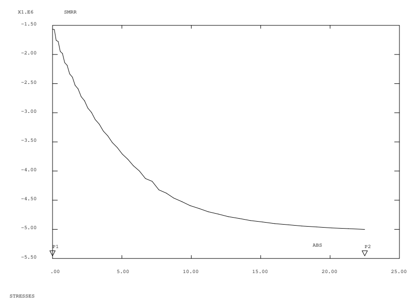
E	= 200.13E06	Young's modulus
NU	= 0.25	Poisson's ratio
RHO	= 0.	density
ALPH	= (5.E-5 / 3.)	thermal expansion coefficient
MOB	= 7500E6	Biot's modulus
COB	= 1.	Biot's coefficient
PERM	= 4.E-12	intrinsic permeability
VISC	= 4.	fluid dynamic viscosity
ALPM	= (19.E-5 / 3.)	coefficient of the pressure-temperature coupling

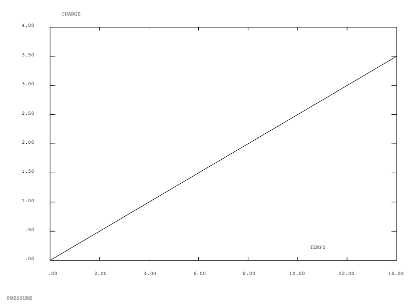


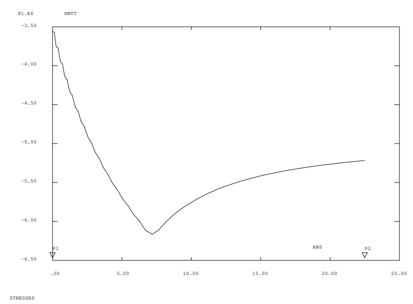
Test pore2 **Results**

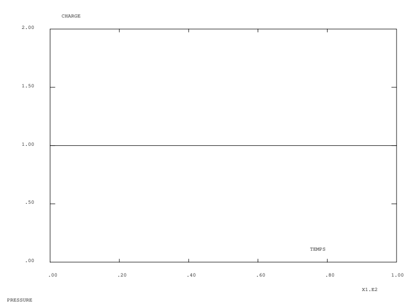
CASTEM FIGURES

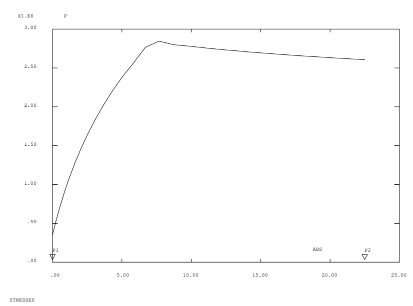












```
*          Test Pore2.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
complet = faux;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
***************************************************
*****************************************************
*                                                   *
*  TEST PORE2                                       *
*                                                   *
*  CYLINDRE EPAIS EN MILIEU POREUX                  *
*  AVEC EFFETS THERMIQUES                           *
*                                                   *
*  REFERENCE : Benchmark INTERCLAY 1.4  Basic Model  *
*                                                   *
*      Le milieu est elastoplastique,               *
*      modele Von-Mises associe                     *
*                                                   *
*      On decharge la contrainte a l'interieur du   *
*      trou puis on impose un champ de temperature  *
*                                                   *
*      On compare avec les valeurs de reference     *
*      obtenues aussi par CASTEM2000 et en accord   *
*      avec les resultats INTERCLAY                 *
*                                                   *
*****************************************************
*
OPTI ECHO 0;
TITRE   ' TEST PORE2  ' ;
GRAPH = 'N';
OPTIO DIME 2 MODEL AXIS   ELEM QUA8  ;
*
*===================================================*
*            geometrie : maillage                   *
*===================================================*
*
* Dimension en metres
*
DENS 0.5;
P1 = 2.5 0. ; P2 = 25. 0. ; Q1 = 2.5 0.5 ;Q2 = 25. 0.5;
PP1 = 5 0. ; QQ1 = 5 0.5 ;
LI1 = P1 D  P2 'DINI' 0.25 'DFIN' 2.5 ;
LI2 =  P2 D Q2 ;
LI3  = INVE ( Q1 D Q2 'DINI' 0.25 'DFIN' 2.5 );
LI4  = Q1 D P1 ;
LI  = LI1  ET LI2  ET LI3  ET LI4  ;
CADRE = DALLER LI1  LI2  LI3  LI4  PLAN ;
SI (NEG GRAPH 'N');
   TRAC CADRE ;
FINSI;
*
*===================================================*
*      modele - materiau - caracteristiques en Pa   *
*                conditions aux limites             *
*===================================================*
*
 MO=MODL CADRE POREUX ELASTIQUE PLASTIQUE
                    DRUCKER_PRAGER ;
*
 MAT=MATR MO YOUNG 200.13E6 NU 0.25 MOB  7500E6 COB 1.
 PERM 4.E-12   VISCO 4.
 ALPH (5.E-5 / 3. )  KF 3000E6   RHOF 0.
 ALPF 40E-5 ALPM (19.E-5 / 3.)  RHO 0.
 ALFA 0.         BETA 0.57735027 K 1.E6
 GAMM 0.         DELT 0.57735027
 ETA  0.         MU   0.57735027 KL 1.E6 H 0. ;
```

```
*
*  on bloque les faces superieure et inferieure
*
BL1=BLOQ UZ  LI1 ;
BL2=BLOQ UZ  LI3 ;
*
BL = BL1 ET BL2 ;
*
*===================================================*
*        1-ere etape  : chargement mecanique        *
*   Initialisation de la table pour appel a PASAPAS  *
*===================================================*
*
*   contraintes initiales
*
ZOZO = MANU CHML MO    SMRZ 0.
   SMRR -5.E6 SMTT -5.E6 SMZZ -5.E6 PRES 2.5E6
   TYPE CONTRAINTES  STRESSES ;
FF = BSIGMA MO ZOZO;
EVT1= EVOL MANU 'TEMPS' (PROG 0       14 )
                'CHARGE' (PROG 1     1 );
 CHA1 = CHAR 'MECA' FF  EVT1 ;
*
*    pression
*
FPI2 = PRESS MASS MO -1.E6   LI4 ;
EVT2= EVOL MANU 'TEMPS' (PROG 0 PAS 1 14 )
                'CHARGE' (PROG 0 PAS 0.25 3.5) ;
CHA2 = CHAR 'MECA' FPI2 EVT2 ;
CHATT = CHA1 ET CHA2 ;
*
TAB1 = TABLE;
TAB1 . 'CONTRAINTES' = TABLE;
TAB1 . 'DEPLACEMENTS' = TABLE;
TAB1.'CONTRAINTES' . 0 = ZOZO;
TAB1.'DEPLACEMENTS' . 0 = MANU CHPO CADRE  1 P 2.5E6 ;
TAB1.'TEMPS_CALCULES' = PROG 0 6  10 14;
TAB1.'MODELE' = MO;
TAB1.'CARACTERISTIQUES' = MAT;
TAB1.'BLOCAGES_MECANIQUES' = BL;
TAB1.'CHARGEMENT' = CHATT;
TAB1.'PRECISION'=5.E-7;
*
PASAPAS TAB1 ;
*
*===================================================*
*           2-eme etape  : temperature              *
*   Initialisation de la table pour appel a PASAPAS  *
*===================================================*
*
TAB2 = TABLE;
TAB2 . 'VARIABLES_INTERNES' = TABLE;
TAB2 . 'DEPLACEMENTS' = TABLE;
TAB2 . 'CONTRAINTES' = TABLE;
SS0 = PECHE TAB1 CONTRAINTES ;
TAB2.'CONTRAINTES' . 0 = SS0;
TAB2.'PRECISION'=5.E-7;
TAB2.'VARIABLES_INTERNES' . 0 =
PECHE TAB1 VARIABLES_INTERNES ;
TAB2.'DEPLACEMENTS' . 0 = PECHE TAB1 DEPLACEMENTS  ;
*
FF = BSIGMA MO SS0 ;
EVT1= EVOL MANU 'TEMPS'  (PROG 0     100)
                'CHARGE' (PROG 1     1 );
CHA1 = CHAR 'MECA' FF EVT1 ;
*
LEFAC = 1. / ( LOG ( 2.5/25.) ) ;
RAD   = COOR 1 CADRE ;
```

```
DT  = NOMC (  LEFAC * ( LOG (RAD / 25. ) ) ) T ;
*
 TABTER = TABLE ;TEMP = TABLE;
 TEMP . 0 = 0.;
 TEMP . 1 = 100.;
 TABTER . 0 = MANU CHPO CADRE 1 T 0. ;
 TABTER . 1 = (DT * 100.)  ;
 CHA2 = CHAR 'T' TEMP TABTER;
*
CHATT = CHA1 ET CHA2;
*
si complet;
tfin = 70.;
sinon;
tfin = 15.;
finsi;
TAB2.'TEMPS_CALCULES' = PROG 0  PAS 5.  tfin; ;
TAB2.'MODELE' = MO;
TAB2.'CARACTERISTIQUES' = MAT;
TAB2.'BLOCAGES_MECANIQUES' = BL;
TAB2.'CHARGEMENT' = CHATT;
*
PASAPAS TAB2 ;
*
*====================================================*
*             affichage des resultats               *
*====================================================*
*
MESS ' RESULTATS TEST PORE2 ';
MESS ' -------------------- ';
SAUT 2 LIGN;
MESS
'   On cherche les contraintes et la pression ';
MESS
'   Elles sont comparees a des valeurs de reference ';
SAUT 1 LIGN;
SAUT 1 LIGN;
*
*----------- r{sultats --------------------------------
*
*    DISTRIBUTION DES CONTRAINTES ET PRESSION
*    POUR T = 70.
*
si complet;
TT = 70. ;
sinon;
TT = 15.;
finsi;
SIGAA = PECHE TAB2 CONTRAINTES  TT;
SIGAB = CHAN 'CHPO' MO SIGAA ;
SIGARR= EVOL CHPO  SIGAB SMRR LI1 ;
SIGATT= EVOL CHPO  SIGAB SMTT LI1 ;
LAPP =  EVOL CHPO  SIGAB PRES LI1 ;
SI (NEG GRAPH 'N');
   DESS SIGARR ; DESS SIGATT ; DESS LAPP ;
FINSI;
EVSIGAR = EXTR SIGARR ORDO 1;
EVSIGAT = EXTR SIGATT ORDO 1;
EVPP    = EXTR LAPP   ORDO 1;
*
*   VALEURS DE REFERENCE  calculees a la date du 1/2/93
*
si complet;
* CONTRAINTE RADIALE
SOLRR = PROG
 -1.59459E+06 -1.56051E+06 -1.78383E+06 -1.76850E+06
 -1.96931E+06 -1.97473E+06 -2.15711E+06 -2.17942E+06
 -2.34668E+06 -2.38270E+06 -2.53753E+06 -2.58467E+06
 -2.72927E+06 -2.78543E+06 -2.92159E+06 -2.98507E+06
 -3.11427E+06 -3.18371E+06 -3.30718E+06 -3.38150E+06
 -3.50030E+06 -3.57863E+06 -3.69361E+06 -3.77540E+06
 -3.88744E+06;
SOLRR = SOLRR ET (PROG
 -3.97178E+06 -4.08114E+06 -4.16891E+06 -4.29345E+06
 -4.34768E+06 -4.48547E+06 -4.54414E+06 -4.62704E+06
 -4.68421E+06 -4.74775E+06 -4.79065E+06 -4.83863E+06
 -4.86963E+06 -4.90481E+06 -4.92612E+06 -4.95084E+06
 -4.96421E+06 -4.98036E+06 -4.98723E+06 -4.99637E+06
 -4.99792E+06 -5.00142E+06 );
*
*  CONTRAINTE TANGENTIELLE
SOLTT = PROG
 -3.59051E+06 -3.55538E+06 -3.77813E+06 -3.76143E+06
 -3.96142E+06 -3.96508E+06 -4.14632E+06 -4.16639E+06
 -4.33216E+06 -4.36546E+06 -4.51847E+06 -4.56247E+06
 -4.70501E+06 -4.75775E+06 -4.89181E+06 -4.95180E+06
 -5.07914E+06 -5.14532E+06 -5.26743E+06 -5.33906E+06
 -5.45722E+06 -5.53370E+06 -5.64905E+06 -5.72971E+06
 -5.84300E+06;
SOLTT = SOLTT ET ( PROG
 -5.92761E+06 -6.03950E+06 -6.12667E+06 -6.24105E+06
 -6.30245E+06 -6.28413E+06 -6.13481E+06 -6.01293E+06
 -5.89978E+06 -5.80418E+06 -5.70545E+06 -5.62098E+06
 -5.53670E+06 -5.46384E+06 -5.39075E+06 -5.32680E+06
 -5.26302E+06 -5.20651E+06 -5.15031E+06 -5.09985E+06
 -5.04987E+06 -5.00444E+06 );
*
*  PRESSION
SOLPP = PROG
  1.36747E+06  1.44034E+06  1.51321E+06  1.58674E+06
  1.66027E+06  1.73387E+06  1.80747E+06  1.88113E+06
  1.95479E+06  2.02829E+06  2.10179E+06  2.17490E+06
  2.24800E+06  2.32049E+06  2.39297E+06  2.46467E+06
  2.53636E+06  2.60709E+06  2.67783E+06  2.74773E+06
  2.81762E+06  2.88621E+06  2.95480E+06  3.02386E+06
  3.09291E+06;
SOLPP = SOLPP ET (PROG
  3.15682E+06  3.22073E+06  3.29927E+06  3.37782E+06
  3.40824E+06  3.43867E+06  3.36939E+06  3.30011E+06
  3.24558E+06  3.19105E+06  3.13296E+06  3.07487E+06
  3.01765E+06  2.96042E+06  2.90299E+06  2.84556E+06
  2.78819E+06  2.73081E+06  2.67343E+06  2.61604E+06
  2.55866E+06  2.50128E+06 );
sinon;
SOLRR = prog
 -1.56503E+06 -1.57213E+06 -1.75778E+06 -1.77784E+06
 -1.94803E+06 -1.98244E+06 -2.14000E+06 -2.18611E+06
 -2.33334E+06 -2.38903E+06 -2.52782E+06 -2.59133E+06
 -2.72321E+06 -2.79313E+06 -2.91938E+06 -2.99454E+06
 -3.11618E+06 -3.19563E+06 -3.31352E+06 -3.39647E+06
 -3.51129E+06 -3.59714E+06 -3.70949E+06 -3.79760E+06
 -3.90787E+06;
solrr = solrr et (prog
 -3.99816E+06 -4.12859E+06 -4.17682E+06 -4.32275E+06
 -4.37916E+06 -4.46316E+06 -4.52542E+06 -4.59310E+06
 -4.64335E+06 -4.69793E+06 -4.73815E+06 -4.78194E+06
 -4.81401E+06 -4.84898E+06 -4.87436E+06 -4.90211E+06
 -4.92201E+06 -4.94386E+06 -4.95928E+06 -4.97629E+06
 -4.98804E+06 -5.00116E+06);
soltt = prog
 -3.56189E+06 -3.56846E+06 -3.75410E+06 -3.77364E+06
 -3.94386E+06 -3.97775E+06 -4.13540E+06 -4.18103E+06
 -4.32842E+06 -4.38367E+06 -4.52270E+06 -4.58583E+06
 -4.71802E+06 -4.78765E+06 -4.91426E+06 -4.98918E+06
 -5.11126E+06 -5.19054E+06 -5.30890E+06 -5.39174E+06
 -5.50708E+06 -5.59285E+06 -5.70571E+06 -5.79390E+06
```

```
 -5.90481E+06;
soltt = soltt et ( prog
 -5.99480E+06 -6.11303E+06 -6.16665E+06 -6.10728E+06
 -6.00175E+06 -5.91312E+06 -5.83171E+06 -5.76581E+06
 -5.69639E+06 -5.63972E+06 -5.58251E+06 -5.53566E+06
 -5.48786E+06 -5.44850E+06 -5.40854E+06 -5.37545E+06
 -5.34188E+06 -5.31389E+06 -5.28556E+06 -5.26174E+06
 -5.23769E+06 -5.21735E+06);

solpp = prog
  3.52566E+05  4.45884E+05  5.39202E+05  6.32610E+05
  7.26018E+05  8.19101E+05  9.12184E+05  1.00502E+06
  1.09786E+06  1.19042E+06  1.28298E+06  1.37529E+06
  1.46759E+06  1.55963E+06  1.65167E+06  1.74345E+06
  1.83523E+06  1.92692E+06  2.01860E+06  2.10969E+06
  2.20078E+06  2.29305E+06  2.38533E+06  2.47274E+06
  2.56016E+06;
solpp = solpp et ( prog
  2.66334E+06  2.76651E+06  2.80560E+06  2.84469E+06
  2.82168E+06  2.79866E+06  2.78895E+06  2.77924E+06
  2.76631E+06  2.75339E+06  2.74124E+06  2.72909E+06
  2.71676E+06  2.70442E+06  2.69213E+06  2.67984E+06
  2.66755E+06  2.65525E+06  2.64295E+06  2.63066E+06
  2.61836E+06  2.60606E+06);

finsi;

*
*   ecarts
*

ERSIGAR =(MAXI ABS ( EVSIGAR - SOLRR ))/
         (MAXI ABS SOLRR);
ERSIGAT =(MAXI ABS ( EVSIGAT - SOLTT ))/
         (MAXI ABS SOLTT);
ERPP    =( MAXI ABS (EVPP - SOLPP))/
         (MAXI ABS SOLPP);
*
SAUT 1 LIGN;
MESS
'Ecart en Contraintes radiales           :' ERSIGAR '%';
SAUT 1 LIGN;
MESS
'Ecart en Contraintes circonferentielles :' ERSIGAT '%';
SAUT 1 LIGN;
MESS
'Ecart en Pressions                      :' ERPP    '%';
*
*
*====================================================*
*                  code fonctionnement               *
*====================================================*
*
* L'ecart maximum entre valeurs de reference et
* calculee doit etre inferieur a 1.d-5
*
SAUT 2 LIGNE;
SI (ERSIGAR <EG 0.001) ;
  SI (ERSIGAT <EG 0.001) ;
    SI (ERPP <EG 0.001) ;
        ERRE 0;
    SINON;
        ERRE 5;
    FINSI;
  SINON;
    ERRE 5;
  FINSI;
SINON;
  ERRE 5;
FINSI;
*
SAUT 1 LIGN;
*
FIN;
```

Test pore2 **Comments**

- The PECHE operator

```
SSO = PECHE TAB1 CONTRAINTES ;
```

This procedure enables the user to retreive for a given time the results of a calculation carried out using the PASAPAS procedure. CONTRAINTES is the requested result index. If no time input (like here), the requested results for the last calculated time will be retreived.

9.3 Test pore3 Description sheet

Test name pore3

Calculation type POROUS-ELASTIC MECHANIC - MODE PLAN CONT **Finite element type**

QUAP

Topic One-dimensionnal consolidation

The structure is a porous elastic piece subjected to a pressure at its upper surface.

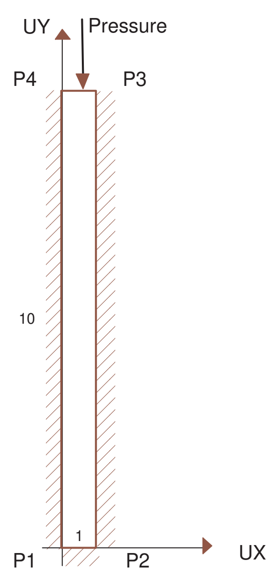
Goal Compare pressures and curves of the vertical displacements with analytical solution.

Reference Problem of Terzaghi

Version 97' customer version

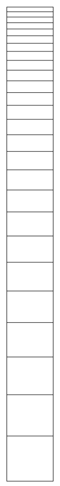
Model description

E	= 7000D6	Young's modulus
NU	= 0.4	Poisson's ratio
RHO	= 0.	density
ALPH	= 0.	thermal expansion coefficient
MOB	= 1.D18	Biot's modulus
COB	= 1.	Biot's coefficient
PERM	= 1.D-9	intrinsic permeability
VISC	= 1.	fluid dynamic viscosity
ALPM	= 0.	coefficient of the pressure-temperature coupling

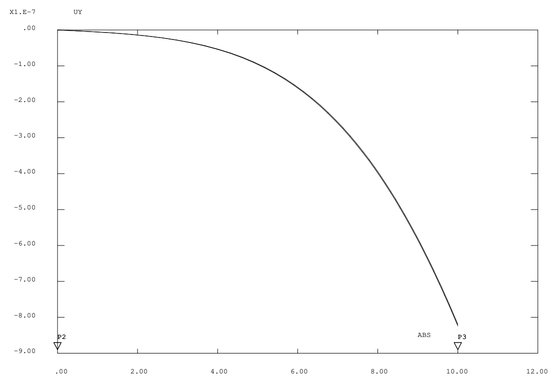


Test pore3 **Results**

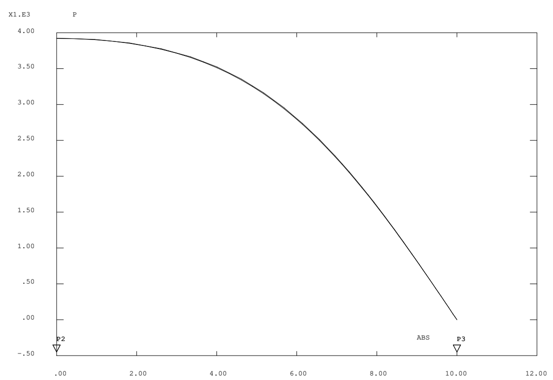
CASTEM FIGURES



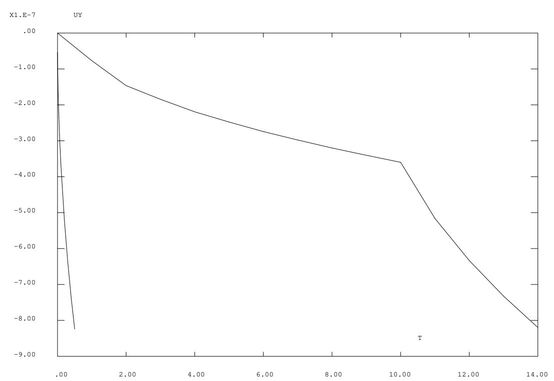
A THICK CYLINDER IN POROUS MIDDLE WITH THERMIC EFFECTS



COMPARISON BETWEEN THE PROFILES OF VERTICAL DISPLACEMENTS



COMPARISON BETWEEN THE PRESSURE PROFILES



COMPARISON BETWEEN THE PROFILES OF VERTICAL DISPLACEMENTS

```
*          Test Pore3.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
complet =faux;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
*************************************************
*****************************************************
*                                                   *
*  TEST PORE3                                       *
*                                                   *
*  CONSOLIDATION UNIDIMENSIONNELLE                  *
*                                                   *
*  REFERENCE : Probleme de Terzaghi                 *
*                                                   *
*      Le milieu est poroélastique                  *
*      On charge à la surface par une pression      *
*      On compare avec la solution analytique       *
*                                                   *
*****************************************************
*
OPTI ECHO 1 ;
TITRE   ' TEST PORE3  ' ;
GRAPH = 'N';
OPTIO DIME 2   ELEM QUA8 MODE PLAN DEFO ;
MOTP = MOTS 'P' ;
LREP = PROG 1.E6  ;
OPTI NORM  MOTP LREP ;
*
*===================================================*
*            geometrie : maillage                   *
*===================================================*
*
DENS 1. ;
*
H = 10. ;H2 = H * H ;
P1 = 0. 0. ; P2 = 1. 0. ; P3 = 1. 10. ; P4 = 0. 10. ;
LI1= P1 D  1 P2 ;
LI2= P2 D    P3 DINI 1. DFIN 0.1 ;
LI3= P3 D 1 P4 ;
LI4= P4 D    P1 DINI 0.1  DFIN 1. ;
LI = LI1 ET LI2 ET LI3 ET LI4 ;
CADRE = DALLER LI1 LI2 LI3 LI4 PLAN ;
SI (NEG GRAPH 'N');
   TRAC CADRE ;
FINSI;
*
*===================================================*
*          modele - materiau                        *
*          rigidite - conditions aux limites        *
*===================================================*
*
MO=MODL CADRE POREUX;
*
LAM = 1.D10 ;
MU = 2.5D9  ;
PK = 1.D-9  ;
*
 MAT=MATR MO   YOUNG  7000D6    NU 0.4    MOB  1.D18
  COB  1.  PERM PK      VISCO 1.
  ALPH 0. KF 1.  RHOF 0. ALPF 0. ALPM 0. RHO 0. ;
BL1=BLOQ UY  LI1 ;
BL2=BLOQ UX (LI2 ET LI4)  ;
BLP = BLOQ P LI3 ;
*
*    on impose une pression sur le dessus
*
```

```
P0 = 4.D3  ;
FPP = PRESSION MASS MO  P0  LI3;
*
*===================================================*
*            consolidation                          *
*===================================================*
*
PER = PERM MO MAT;
ev = evol manu t ( prog 0 1.D-5 1000) F  ( PROG 0 1 1);
CHA = CHAR 'MECA' FPP EV ;
si complet;
LETEMP= PROG 0 PAS  1.D-2 1.D-1 PAS 1.D-1 1.D0
        PAS 5.D-1 1.D1;
sinon;
letemp = prog 0 PAS  1.D-2 1.D-1 PAS 1.D-1 5.d-1;
finsi;

NBT = DIME LETEMP ;
TFINAL = EXTR LETEMP NBT;

*
*====================================================
*             remplissage de la table
*====================================================
*
TAB1 = TABLE ;
TAB1.'MODELE' = MO;
TAB1.'CARACTERISTIQUES' = MAT;
TAB1.'BLOCAGES_MECANIQUES' = BL1 ET BL2 ET BLP;
TAB1.'TEMPS_CALCULES' = LETEMP;
TAB1.'CHARGEMENT' = CHA;
TAB1.'CONSOLIDATION' = VRAI;
TAB1.'PERMEABILITE' = PER ;
*
*  appel a PASAPAS
*
PASAPAS TAB1;
*
*===================================================*
*            affichage des resultats                *
*===================================================*
*
MESS ' RESULTATS TEST PORE3 ';
MESS ' -------------------- ';
SAUT 2 LIGN;
MESS
'  On cherche le deplacement vertical et la pression ';
MESS
'  on compare aux solutions analytiques  ';
SAUT 1 LIGN;
SAUT 1 LIGN;
*
TTA = INDEX ( TAB1.DEPLACEMENTS);
NT = DIME TTA ;
LIST NT;
IT = 0 ;
  REPETER BOPI NT;
    IT = IT + 1 ;
    LET = TTA . IT;
    DD= TAB1.DEPLACEMENTS.LET;
SI ( EGA IT 1 ) ;
   PRT = PROG LET ;
   PRD = PROG ( EXTR DD UY P3);
   EV = EVOL CHPO DD P LI2 ;
SINON;
   PRT = PRT ET (PROG LET) ;
   PRD = PRD ET (PROG ( EXTR DD UY P3)) ;
   EV = EV ET (EVOL CHPO DD P LI2) ;
```

```
FINSI;
   FIN BOPI ;
FV = EVOL ROSE MANU 'T' PRT 'UY'  PRD;
*
DD= PECHE TAB1 DEPLACEMENTS ;
EV1= EVOL ROSE CHPO DD P LI2 ;
EVD1= EVOL ROSE CHPO DD UY LI2 ;
*
*=====================================================*
*  determination de la solution analytique
*=====================================================*
*
X = COOR 2 LI2;
XX =  ( MANU CHPO LI2 1 SCAL 1 ) - (X / H) ;
XX = NOMC XX P ;
FACT =    (LAM + ( 2. * MU )) * ( PK / H2 ) ;
FACU = -1. *  ( (LAM + ( 2. * MU )) / ( P0 * H ) ) ;
PI =  3.141592654D0;
PI2 = PI / 2.D0 ;
DEGPI = 180. / PI ;
*
ITT = 0 ;
REPE BOUT NBT;
   ITT = ITT + 1 ;
   T=  EXTR LETEMP ITT ;
   PP = MANU CHPO LI2 1 P 0. ;
   UU = X / H ;
   UU = NOMC UU P ;
   TT = FACT * T;
   N =  0;
si complet ;
nfoi = 50;
sinon;
nfoi = 10;
finsi;
  REPE BOU nfoi;
   M = PI2 * ( (2. * N ) + 1. ) ;
   M2 = M*M ;
   EX = EXP  ( ( -1. * M2) * TT ) ;
   SSI =  SIN ( (M *  DEGPI)  * XX )  ;
   CCO =  COS ( (M *  DEGPI)  * XX )  ;
   PP = PP + ( ( ( 2. / M ) * EX) * SSI) ;
   UU = UU - ( ( ( 2. / M2) * EX) * CCO) ;
   N=N + 1 ;
  FIN BOU ;
   PP = PP * P0 ;
   UU = UU / FACU ;
SI ( EGA ITT 1 ) ;
   PGT = PROG T ;
   PGU = PROG ( EXTR UU 'P' P3) ;
SINON ;
   PGT = PGT ET (PROG T) ;
   PGU = PGU ET (PROG ( EXTR UU 'P' P3)) ;
FINSI;
*
SI ( EGA ITT NBT ) ;
   EV2= EVOL TURQ CHPO PP 'P' LI2;
 SI (NEG GRAPH 'N');
   TITRE ' COMPARAISON DES PROFILS DE PRESSION ' ;
   DESS (EV1 ET EV2 );
 FINSI;
  EVD2= EVOL TURQ CHPO UU 'P' LI2;
 SI (NEG GRAPH 'N');
   TITRE
   ' COMPARAISON DES PROFILS DE DEPLACEMENT VERTICAL';
   DESS (EVD1 ET EVD2 );
 FINSI;
FINSI;
FIN BOUT;
FVT= EVOL TURQ MANU 'T' PGT 'UY'  PGU;
SI (NEG GRAPH 'N');
 DESS ( FV ET FVT) ;
FINSI;
*
*   ecarts
*
EVY1 = EXTR EV1 ORDO ;
EVY2 = EXTR EV2 ORDO ;
EVDY1 = EXTR EVD1 ORDO ;
EVDY2 = EXTR EVD2 ORDO ;
ERPP    =(MAXI ABS ( EVY1 - EVY2   ))/(MAXI ABS EVY2 );
ERDUY   =( MAXI ABS (EVDY1 - EVDY2 ))/(MAXI ABS EVDY2);
*
*
SAUT 1 LIGN;
MESS
'Ecart en pression                       :' ERPP    '%';
SAUT 1 LIGN;
MESS
'Ecart en profil de deplacement vertical :' ERDUY   '%';
*
*
*=====================================================*
*                  code fonctionnement                *
*=====================================================*
*
* L'ecart maximum entre valeurs de reference et
* calculee doit etre inferieur a 1.d-3
*
SAUT 2 LIGNE;
SI (ERDUY   <EG 0.1) ;
    SI (ERPP <EG 0.1) ;
        ERRE 0;
    SINON;
        ERRE 5;
    FINSI;
SINON;
  ERRE 5;
FINSI;
*
SAUT 1 LIGN;
*
FIN ;
```

Test pore3 **Comments**

1. The POREUX formulation

```
MO=MODE CADRE POREUX ;
LAM = 1.D10  ;
MU  = 2.5D9  ;
PK  = 1.D-9  ;
MAT=MATE MO YOUNG 7000D6  NU 0.4   MOB  1.D18
    COB  1. PERM PK       VISCO 1. ALPH 0.  KF 1.
    RHOF 0. ALPF 0.       ALPM 0.  RHO 0. ;
```

   This procedure is to create a porous model and to give the material data.

2. The MOTS operator

```
MOTP = MOTS P ;
```

   The MOTS operator creates a list of 4-character words.

3. The OPTI NORM operator

```
OPTI NORM  MOTP LREP ;
```

   The NORM option in the OPTI operator allows to normalize variables in view of factorization. It is required that a LISTMOT containing the initial unknowns to be normalized be given as well as the LISTREEL providing the standardization factor. We may also ask for automatic standardization ('AUTO') but this may take some time.

4. The INDEX operator

```
TTA = INDEX ( TAB1.DEPLACEMENTS) ;
```

   The INDEX operator enables the user to get all the indices of a table. 'TAB1.DEPLACEMENTS' and 'TTA' are respectively the table which indices are requested and the generated object containing the indice (TABLE type).

Chapter 10

METALLURGY OF POWDERS

10.1 Test poudre1 Description sheet

Test name poudre1

Calculation type POWDER COMPACTION - 2D MODE AXIS

Finite element type QUA8

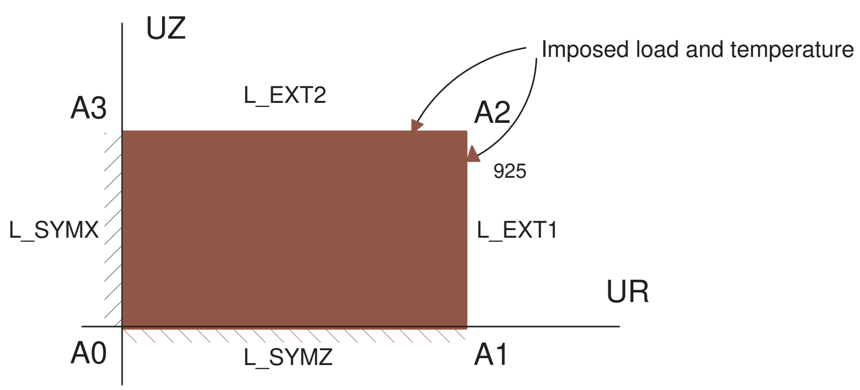
Topic Viscoplastic model of Abouaf for consolidation of powders.

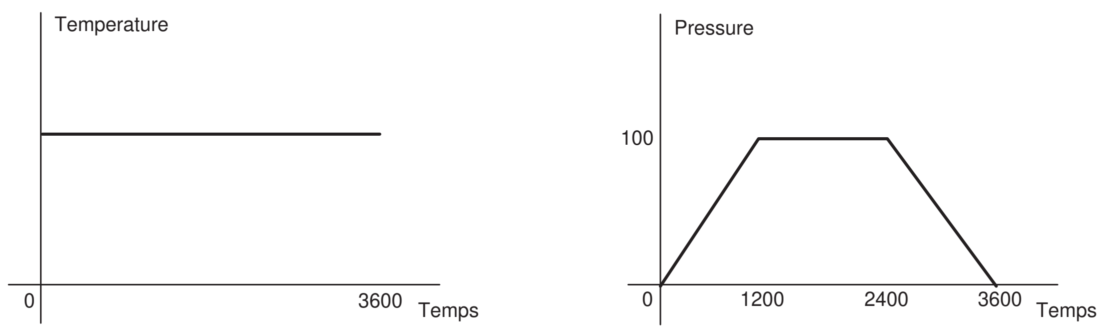
The structure is a cylinder consolidated by pressure load. The initial density is assumed uniform, whereas the final density is obtained with analytical calculation. The cylinder is also subjected to a field of temperature. It follows a viscoplastic law according to the model of Abouaf. **Goal** Test the model of Abouaf: poudre_A.

Version 97' customer version

Model description

$$
\begin{aligned}
E &= 118.0\ 10^3\ \text{Pa} \\
NU &= 0.33
\end{aligned}
$$



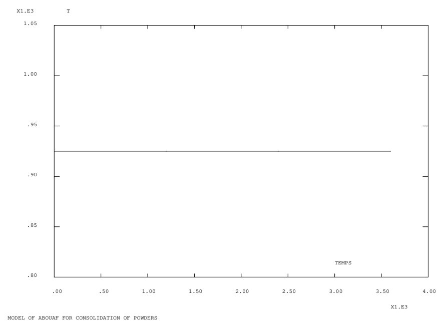


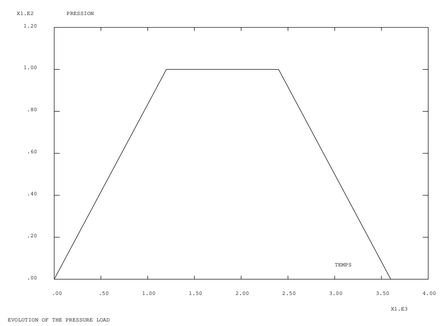
Test poudre1 **Results**

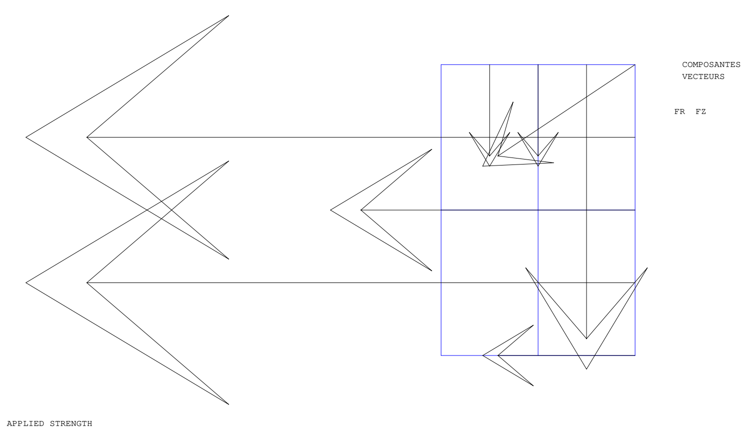
RESULTS: Displacements for an isotropic densification

	UR	UZ
Analytical results	$-1.25297E-03$	$-1.88229E-03$
Castem results	$-1.25292E-03$	$-1.88222E-03$

CASTEM FIGURES







```
complet = faux ;
*
**************************************************
*  pour calcul complet mettre complet à : vrai;
****************************************************
*
****************************************************************
*
*    Cas test pour la loi poudre_A
*
*    par Christophe DELLIS   (CEREM)
*
****************************************************************
*  cas isotrope
*  un cylindre est densifié par mise en pression
*  la densite initial est uniforme la densite finale
*  est connue analytiquement
*
*opti echo 0 ;
*
option dime 2 elem qua8 mode axis ;
*
*    parametres du maillage
*
rayon_1 = 10.0 ;
*
haute_1 = 15.0 ;
*
* definition des coordonnees des points
*
xa0 = 0.0 ;
ya0 = 0.0 ;
*
xa1 = rayon_1 ;
ya1 = ya0 ;
*
xa2 = xa1 ;
ya2 = ya1 + haute_1 ;
*
xa3 = 0.0 ;
ya3 = ya2 ;
*
* definition des densites
*
dena0 = 10.0 ;
dena1 = 10.0 ;
dena2 = 10.0 ;
dena3 = 10.0 ;
*
* definition des points
*
densite dena0 ;
a0 = xa0 ya0 ;
*
densite dena1 ;
a1 = xa1 ya1 ;
*
densite dena2 ;
a2 = xa2 ya2 ;
*
densite dena3 ;
a3 = xa3 ya3 ;
*
* definition des lignes
*
la0a1 = a0 droi 2 a1 ;
la1a2 = a1 droi 2 a2 ;
la2a3 = a2 droi 2 a3 ;
```

```
la3a0 = a3 droi 2 a0 ;
*
la3a2 = inve la2a3 ;
la2a1 = inve la1a2 ;
*
* lignes pour les conditions limites
*
l_symz = la0a1 ;
l_symx = la3a0 ;
l_ext1 = la1a2 ;
l_ext2 = la2a3 ;
*
* definition des surfaces
*
elmat1 = surf ( la0a1 et la1a2 et la2a3 et la3a0 ) plan ;
*
piece_1 = coul elmat1 bleu ;
*
tot = piece_1 ;
elim (tot et a2) 0.001 ;
*
*
*************************************************************
*                                                           *
*       calcul                                              *
*                                                           *
*************************************************************
*
T0TEMPE = 925.0 ;
T1TEMPE = 925.0 ;
T0TEMPS = 0.0 ;
T1TEMPS = 1200.0 ;
T2TEMPS = 2400.0 ;
T3TEMPS = 3600.0 ;
*
*******************************************
* Champ-point de temperature
*******************************************
*
clth0001 = BLOQ 'T' (l_ext1 et l_ext2) ;
p_bloth1 = DEPI clth0001 1. ;
p_temps1 = prog T0TEMPS T1TEMPS T2TEMPS T3TEMPS ;
p_theta1 = prog T0TEMPE T1TEMPE T1TEMPE T0TEMPE ;
ev_ther1 = EVOL MANU TEMPS p_temps1 'T' p_theta1 ;
ch_ther1 = CHAR TIMP p_bloth1 ev_ther1 ;
*
*************************************************************
*                                                           *
*       materiau                                            *
*                                                           *
*************************************************************
P1MODEL = MODE piece_1 MECANIQUE ELASTIQUE
VISCOPLASTIQUE POUDRE_A CONS DEUX;
*
V1YOUNG = 118.0E+03 ;
V1NU = 0.33 ;
V1ALPHA = 10.0E-10 ;
V1A = 7.76E-15 ;
V1N = 4.55 ;
V1QSRT = 0.0 ;
V1RHOR = 0.68 ;
V1F0 = 0.1098 ;
V1F1 = 1.0168 ;
V1F2 = -0.2591 ;
V1F3 = 0.0 ;
V1F4 = 0.0 ;
V1F5 = 0.66 ;
V1C0 = 2.10851 ;
```

```
V1C1 = 1.21767 ;
V1C2 = -0.43081 ;
V1C3 = 0.0 ;
V1C4 = 0.0 ;
V1C5 = 50.0 ;
*
P1MATER = MATER P1MODEL YOUNG V1YOUNG NU V1NU ALPHA V1ALPHA
 A V1A N V1N QSRT V1QSRT F0 V1F0 F1 V1F1 F2 V1F2 F3 V1F3
 F4 V1F4 F5 V1F5 C0 V1C0 C1 V1C1 C2 V1C2 C3 V1C3 C4 V1C4
 C5 V1C5 RHOR V1RHOR ;
*
P2MODEL = MODE piece_1 THERMIQUE ISOTROPE CONS UN ;
*
P2MATER = MATER P2MODEL 'C' 460 'K' 14.6 'RHO' 4.54 ;
*
************************************************************
*                                                          *
*          chargement                                      *
*                                                          *
************************************************************
*
p0bloca = bloque uz l_symz ;
p1bloca = bloque ur l_symx ;
*
p_bloca = p0bloca et p1bloca ;
*
t0pres = 0.0 ;
t1pres = 100.0 ;
*
li_tps = prog t0temps t1temps t2temps t3temps ;
li_pres = prog t0pres t1pres t1pres t0pres ;
ev_pres = evol manu temps li_tps pression li_pres ;
*dess ev_pres titr 'Evolution du chargement en pression' nclk ;
*
p2force = pres mass p1model 1.0 l_ext1 ;
p3force = pres mass p1model 1.0 l_ext2 ;
titr 'Force appliquees' ;
vec1 = vect (p2force et p3force) 'FR' 'FZ' 0.1;
*trac vec1 piece_1 nclk ;
*
p2charg = char p2force ev_pres 'MECA';
p3charg = char p3force ev_pres 'MECA';
*
p_charg = p2charg et p3charg ;
*
************************************************************
*                                                          *
*        calcul                                            *
*                                                          *
************************************************************
*
p_model = p1model et p2model ;
p_mater = p1mater et p2mater ;
*p_model = p1model ;
*p_mater = p1mater ;
*
* Initialisation de la table pour la procedure PASAPAS
*
TAB1 = TABLE ;
TAB1 . 'CHARGEMENT' = (p_charg et ch_ther1) ;
TAB1 . 'MODELE' = p_model ;
TAB1 . 'CARACTERISTIQUES' = p_mater ;
TAB1 . 'BLOCAGES_MECANIQUES' = p_bloca ;
TAB1 . 'BLOCAGES_THERMIQUES' = clth0001 ;
si complet;
tfin = 2400.;
sinon;
tfin = 400.;
finsi;

TAB1 . 'TEMPS_CALCULES' = PROG 0.0
        PAS 50.0 tfin ;
TAB1 . 'TEMPS_SAUVES' = PROG tfin ;
TAB1 . 'MAXITERATION' = 49 ;
TAB1 . 'MAXISOUSPAS' = 500 ;
TAB1 . 'MOVA' = 'VHOR' ;
TAB1 . 'PRECISION' = 1.0E-4 ;
TAB1 . 'TALPHA_REFERENCE' = 20.0 ;
TAB1 . 'ACCELERATION' = 2 ;
TAB1 . 'PROCEDURE_THERMIQUE' = LINEAIRE ;
*
PASAPAS TAB1 ;
*
************************************************************
*                                                          *
*     Test de validation                                   *
*                                                          *
************************************************************
*
ind1 = index (TAB1 . 'VARIABLES_INTERNES') ;
nb1 = dime ind1 ;
*
*   densite finale
*
vm1 = TAB1 . 'VARIABLES_INTERNES' .(ind1.nb1) ;
vm2 = exco VHOR vm1 ;
vm3 = redu vm2 piece_1 ;
rhof1 = maxi vm3 ;
*
* Calcul theorique analytique de la densite finale :
*
si complet;
rhof2 = 0.7418622 ;
sinon;
rhof2 = 0.68006 ;
finsi;
*
* Lecture des deplacements du noeud au coin de l'echantillon :
*
ind2 = index (TAB1 . 'DEPLACEMENTS' ) ;
nb2 = dime ind2 ;
ur1 = (extr TAB1 .'DEPLACEMENTS'.(ind2.nb2) ur a2) ;
uz1 = (extr TAB1 .'DEPLACEMENTS'.(ind2.nb2) uz a2) ;
*
* Calcul analytique des deplacements pour une densification isotrope :
*
si complet;
ur2 = rayon_1 * ((v1rhor/rhof1)**(1./3.) - 1.) ;
uz2 = haute_1 * ((v1rhor/rhof1)**(1./3.) - 1.) ;
sinon;
ur2 = -1.25297E-03  ;
uz2 = -1.88229E-03  ;
finsi;
list ur2 ;list ur1;
list uz2 ; list uz1;
*
* Calcul des erreurs sur les deplacements et la densite :
*
erho1 = abs ((rhof1 - rhof2 )/rhof2);
eur1 = abs ((ur1 - ur2) / ur2) ;
euz1 = abs ((uz1 - uz2) / uz2) ;
*
* Ecriture des erreurs a l'ecran :
*
opti echo 0 ;
sauter 3 lignes ;
```

```
mess 'Calcul en isotrope :' ;
mess 'Erreur sur le deplacement radial :    '  eur1 ;
mess 'Erreur sur le deplacement vertical :  '  euz1 ;
mess 'Erreur sur la densite finale:         '  erho1 ;
sauter 3 lignes ;

*
si ( (eur1 + euz1 + erho1 ) >eg 0.05 ) ;
   mess '---------RESULTATS INCORRECTS-------------' ;
   sauter 3 lignes ;
   erreur 5;
sinon ;
   mess '---------RESULTATS CORRECTS-------------' ;
   sauter 3 lignes ;
finsi ;
*
fin ;

*********************************************************************
```

Test poudre1 **Comments**

1. ABOUAF model for consolidation of powders: POUDRE_A

```
P1MODEL = MODE PIECE_1 MECANIQUE ELASTIQUE
~~~VISCOPLASTIQUE POUDRE_A CONS DEUX ;
P1MATER = MATE P1MODEL YOUNG V1YOUNG NU V1NU ALPHA V1ALPHA
~~~A V1A N V1N QSRT V1QSRT F0 V1F0 F1 V1F1 F2 V1F2 F3 V1F3
~~~F4 V1F4 F5 V1F5 C0 V1C0 C1 V1C1 C2 V1C2 C3 V1C3 C4 V1C4
~~~C5 V1C5 RHOR V1RHOR ;
```

The ABOUAF model is a viscoplastic model for consolidation of powders.
The constitutive equations of Abouaf model are:

Seq = (f*I1**2+3/2*c*J2**2)**0.5
devp/dt = rho*A*exp(-Q/RT)*Seq**(n-1)*(f*I1*d+3/2*c*s)
tr(devp/dt) = -(drho/dt)/rho

With the following notations:

S	stress tensor
s	deviatorie stress tensor
d	unit tensor
Seq	equivalent stress
I1	first invariant of stress tensor
J2	second invariant of deviatoric stress tensor
rho	relative density of porous material
f	function of relative density
c	function of relative density
evp	viscoplastic deformation
t	time
T	temperature

We should put several data in the MATE operator:

A	coefficient of creep law
N	power creep law exponent
QSRT	activation energy
F0 to F5	parameters defining the f function
C0 to C5	parameters defining the c fonction
RHOR	initial relative density of the material

2. Isotropic thermal model

```
P2MODEL = MODE PIECE_1 THERMIQUE ISOTROPE CONS UN ;
P2MATER = MATE P2MODEL 'C' 460 'K' 14.6 'RHO' 4.54 ;
```

With the isotropic thermal model, we have to specify the specific heat (C), the thermal conductivity (K) and the density (RHO). The word CONS indicate the number of component. It allows to associate in the same calculation several models to a same mesh object.

Chapter 11

JOINT

11.1 Test joi22 Description sheet

Test name joi22

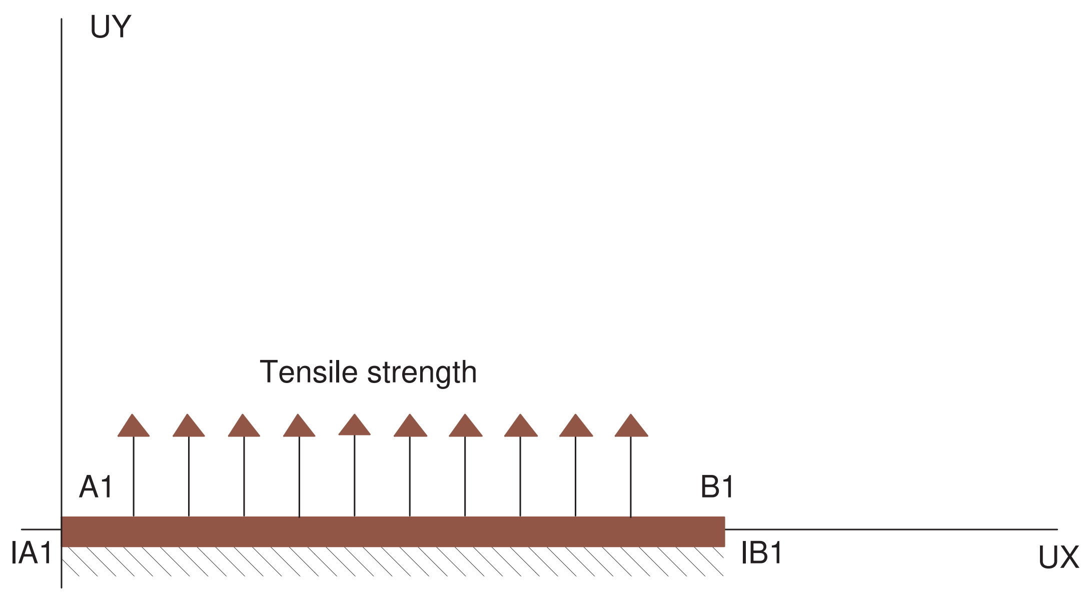
Calculation type ISOTROPIC ELASTIC MECHANIC **Finite element type** SEG2, JOI2

Topic Tensile test on a 2D-joint.

The structure is a 2D-joint subjected to a tensile strength on its upper surface, in the direction perpendicular to its plan. It is embedded from its lower surface. **Goal** Test the JOI2 element.

Version 97' customer version

Model description



Test joi22 **Results**

RESULTS Analytical solution:

From the principle of virtual work, we can show for a one-axis tensile test that the delta displacement at each node is equal to:

$$\text{delta} = \frac{F}{k * L * l}$$

where:
F is the strength
k is the stiffness
L is the length of the joint element
l is the width of the joint element

Numerical application:

F = 100000.0 N
k = 4.2E10 N/m3
L = 2.0 m
l = 1.0 m

Delta = 100000 / (4.2E10 * 2.0 * 1.0)
= 1.190476E-6

Castem solution:

Point	Unknown	Point	Unknown	Point	Unknown	Point	Unknown
5	0.00000E0	6	0.00000E0	7	2.50000E4	8	2.50000E-4
9	0.000E0	10	2.50000E4	11	2.50000E4	12	2.50000E-4

Point	*UX*	*UY*
1	$0.00000E0$	$1.19048E-6$
2	$0.00000E0$	$1.19048E-6$
3	$0.00000E0$	$-2.11615E-22$
4	$0.000000E0$	$-1.05736E-22$

Strain calculation	
First component	
point 1 :	$0.000E0$
point 2 :	$0.0000E0$
Second component	
point 1 :	$-1.190E-6$
point 2 :	$-1.190E-6$

Stress calculation	
First component	
point 1 :	$0.000E0$
point 2 :	$0.0000E0$
Second component	
point 1 :	$-5.000E4$
point 2 :	$-5.000E4$

```
*          Test Joi22.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTI ECHO 0 ;
SAUT PAGE ;
MESS'!=============================================!';
MESS'!                                             !';
MESS'!                  TEST JOI22                 !';
MESS'!                                             !';
MESS'!      ESSAI DE TRACTION SUR UN JOINT 2D      !';
MESS'!                                             !';
MESS'!  Un joint 2D JOI2 a son bord inferieur      !';
MESS'!  encastre.                                  !';
MESS'!  Son bord superieur est libre. Un effort de !';
MESS'!  traction est exercé sur son bord superieur. !';
MESS'!                                             !';
MESS'!                                             !';
MESS'!  Solution analytique :                      !';
MESS'!                                             !';
MESS'!     En ecrivant le principe des travaux     !';
MESS'!  virtuels,on montre que le deplacement delta !';
MESS'!  en chaque noeud est egal, dans le cas d un  !';
MESS'!  essai de traction uniaxial, a :            !';
MESS'!                       F                     !';
MESS'!        delta = ----------------             !';
MESS'!                    k * L * l                !';
MESS'!  ou                                         !';
MESS'!    F = force totale exercee sur la structure !';
MESS'!    k = raideur                              !';
MESS'!    L = longueur de l element joint          !';
MESS'!    l = largeur de l element joint           !';
MESS'!                                             !';
MESS'!  Application numerique :                    !';
MESS'!                                             !';
MESS'!    F = 100000.0                             !';
MESS'!    k = 4.2E10                               !';
MESS'!    L = 2.0                                  !';
MESS'!    l = 1.0                                  !';
MESS'!                                             !';
MESS'!    Delta   =  100000 / (4.2E10 * 2.0 * 1.0)  !';
MESS'!            =  1.19047619E-6                 !';
MESS'!                                             !';
MESS'!=============================================!';
OPTION DIME 2 ;
OPTION ELEM SEG2 MODE PLAN CONT  ;
*
SOLANA = 1.190476E-6 ;
SOLNUL = 0.0 ;
*
* ------- DEFINITION DE LA SURFACE TOP DU JOINT -------
*
A1 = 0.00 0.00 ;
B1 = 2.00 0.00 ;
*
* ----------          MAILLAGE          ----------
*
H1 = A1 DROIT 1 B1 ;
*
L1 = H1 ;
*
* ------ DEFINITION DE LA SURFACE BOT DU JOINT --------
*
IA1 = 0.00 0.00 ;
IB1 = 2.00 0.00 ;
*
* ----------          MAILLAGE          ----------
*
IH1 = IA1 DROIT 1 IB1 ;
```

```
*
IL1 = IH1 ;
*
* ----------   CREATION DU JOINT JOI2   ----------
*
OPTION ELEM RAC2 ;
VOL = RACCORD 0.00001 L1 IL1 ;
*
* --------- DEFINITION DES CONDITIONS LIMITES ---------
*
CL11 = BLOQ IA1 UX ;
CL12 = BLOQ IA1 UY ;
CL1 = CL11 ET CL12 ;
*
CL21 = BLOQ IB1 UX ;
CL22 = BLOQ IB1 UY ;
CL2 = CL21 ET CL22 ;
*
CL = CL1 ET CL2 ;
*
* -----    DEFINITION DU MODELE DU JOINT      ---------
*
MOD1 =MODL VOL 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE' JOI2;
MA1 = MATR MOD1 KS 4.2E08 KN 4.2E10 ;
*
* -------------    MATRICE DE HOOK         ------------
*
HOO1 = HOOKE MOD1 MA1 ;
******LIST HOO1 ;
*
* ----------       MATRICE DE RIGIDITE       ---------
*
******RI1 = RIGI MOD1 MA1 ;
RI1 = RIGI MOD1 HOO1 ;
RI2 = RI1 ET CL ;
*
* ----------        FORCE DE TRACTION        ---------
*
FO1 = FORCE ( 0. 100000.0 ) L1 ;
*
* ---------            RESOLUTION           ----------
*
RE = RESO RI2 FO1 ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS ' Solution Analytique :'                         ;
MESS '                                              ' ;
MESS '    UX =' SOLNUL  ;
MESS '    UY =' SOLANA  ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS ' Solution Calculee :'                           ;
MESS '                                              ' ;
LIST RE ;
*
* ----------   CODE DE FONCTIONNEMENT      ----------
*
DEPA1 = EXTR RE UY A1 ;
RESI = ABS( (DEPA1-SOLANA)/SOLANA ) ;
*
SI (RESI <EG 1E-4 ) ;
   ERRE  0 ;
SINO;
   ERRE  5 ;
FINSI ;
*
* ----------    CALCUL DES DEFORMATIONS    ----------
```

```
*
EPS1 = EPSI MOD1 RE ;
LIST EPS1 ;
*
* ----------    CALCUL DES CONTRAINTES       ----------
*
SIG1 = SIGMA MOD1 MA1 RE ;
LIST SIG1 ;
*
FIN ;
```

Test joi22 **Comments**

1. JOINT operator

```
MOD1 = MODE VOL MECANIQUE ELASTIQUE ISOTROPE JOI2 ;
MA1 = MATE MOD1 KS 4.2E08 KN 4.2E10 ;
```

The JOI2 element is used in the modelling of a 2D-joint. This joint is an element with four nodes associated with the geometrical support RAC2. Within the MATE operator we should specify the shear and the normal stiffnesses, respectively KS and KN.

2. RACCORD operator

```
OPTI ELEM RAC2 ;
VOL = RACCORD 0.00001 L1 IL1 ;
```

The RACCORD operator enables the user to create an ordinary joint element. It generates a line VOL (MAILLAGE type) composed of double points that connects the objects L1 and IL1 (MAILLAGE type). The objects L1 and IL1 may be segments of 2 or 3 points. An element is created between an element of the L1 object and an element of the IL1 object; the distance between their points is smaller than a distance criterion (FLOTTANT type: 0.00001).
For the creation of a 2D joint element JOI2 (or JOI3) L1 and IL1 represent respectivelly lines 1 and 2 of the element.These lines should be read in the same direction. This direction is that of line 1. It is defined by considering a vector n normal to this line. When it is orientated positivelly in the direction of the opening of the joint element, n must necessarilly lie on the right handside of the segment going from node 1 to node 2 of line 1. (We define the opening of the element as the separation movement of line 1 from line 2 when line 2 is held fixed).

3. HOOKE operator

```
HOO1 = HOOKE MOD1 MA1 ;
```

The HOOKE operator constructs the field of HOOKE's matrix from a field of geometrical and material properties MA1 (MCHAML type, CARACTERISTIQUES subtype). Here HOO1 is a field by element of Hooke's matrices (MCHAML type, MATRICE DE HOOKE subtype).
MA1 is required only for the types of elements the geometry of wich cannot be deduced from the mesh, for instance, beam, pipe, or shell elements.

4. EPSI and SIGMA operators

```
EPS1 = EPSI MOD1 RE ;            SIG1 = SIGMA MOD1 MA1 RE ;
```

The EPSI operator enables to calculate a strain field either from a displacement field, according to usual expressions with or without second order terms (1-st syntax), or by taking the natural logarithm of a symmetric gradient field (2-nd syntax). This second possibility is now available only for the massive formulation.
For some elements, (beams, pipes, thin shells, with or without transverse shear) it describes generalized strains, i.e. membrane-type strains and curvature variations. For the joint elements, it describes relative displacements. The strains for the solid elements are computed in the general basis, those for the shell, plate, and beam elements are computed in the local axes.
Here MOD1 is a model object (MMODEL type), GRAD1 is a symmetrical gradient field (MCHAML type, GRADIENT subtype) and EPS1 is the generated displacement field (MCHAML type, DEFORMATIONS subtype).

The SIGMA operator calculates a stress field from a displacement field. It is assumed that the material has an elastic linear behaviour and that there is no initial strain. For some elements, it describes forces, (bars, beams, pipes), for others, it describes generalized stresses (thin shells). The stresses are calculated in the general basis for the solid elements, and in the local axes for the shell, plate and beam elements. Here SIG1 is the generated stress field (MCHAML type, CONTRAINTES subtype).

11.2 Test joi41 Description sheet

Test name joi41

Calculation type ISOTROPIC ELASTIC MECHANICS **Finite element type** QUA4, CUB8,JOI4

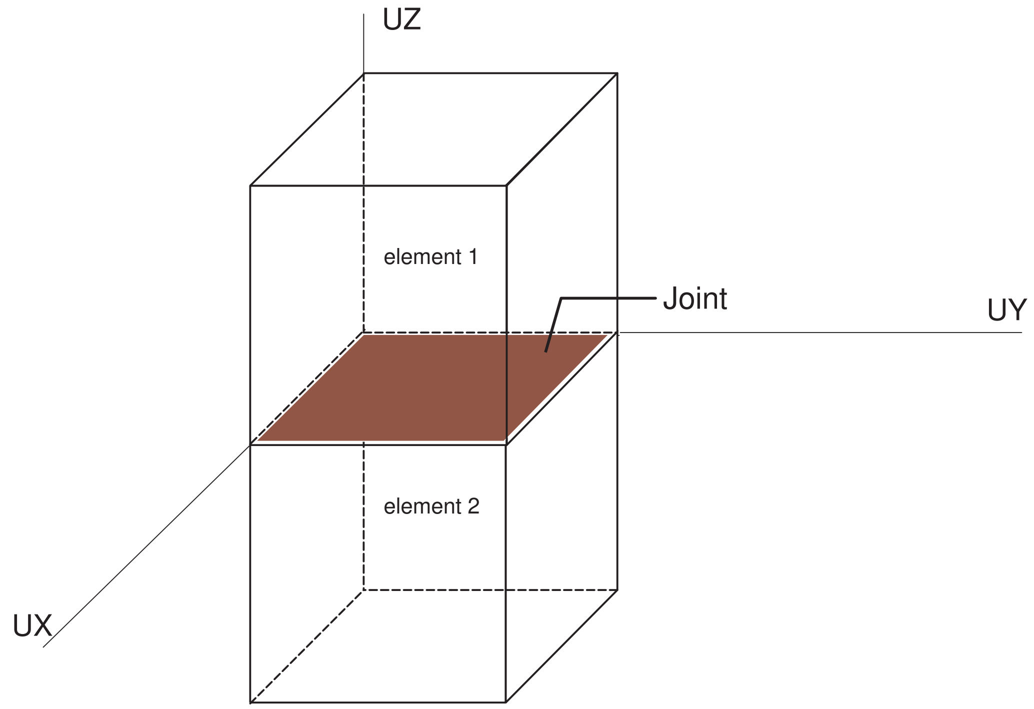
Topic Two superposed cubes with a JOI4 joint in their intersection
We study the effects of joints with the help of two examples: cubes with joint and cubes without joint.

Goal Calculate the effect of a JOI4 joint on displacements, strains and stresses.

Version 97' customer version

Model description

$$E = 2.1\ 10^{11}\ \text{Pa}$$
$$NU = 0.3$$



```
*          Test Joi41.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTI ECHO 0;
SAUT PAGE;
*
*******************************************************
*                                                   *
*                       TEST JOI41                  *
*                                                   *
*       2 CUBES SUPERPOSES AVEC UN JOINT JOI4       *
*           AU MILIEU ( JOI4 3D ISOTROPE )          *
*                                                   *
*  Deux CUB8 sont superposes. On place un joint JOI4  *
*  entre les 2 cubes. Les noeuds de la surface infe-  *
*  -rieure du cube du bas sont encastres. Une force   *
*  de traction verticale (i.e. suivant z) est exercee *
*  sur la surface superieure du cube du haut.        *
*  Deux cas sont etudies :                           *
*   - 1 / On suppose que le joint n'existe pas entre  *
*         les deux cubes.                            *
*   - 2 / On suppose que le joint existe et a une     *
*         raideur normale tres grande par rapport a   *
*         celle de chacun des cubes.                 *
*  Les deplacements UZ des noeuds des deux cubes      *
*  doivent etre les memes dans les deux cas.         *
*  Raideur de chacun des cubes :                     *
*       sigma = force / A        A = section         *
*  or   force = k * delta        k = raideur         *
*                                delta = deplacement *
*  de + sigma = E * epsilon      E = Young           *
*  ou epsilon = delta / L        L = hauteur du cube *
*  ---> k     = E * A / L                            *
*  ---> k     = 4.2 E11                              *
*  On choisit pour raideur du joint, la valeur       *
*       k     = 4.2 E15                              *
*                                                   *
*******************************************************
*
TEMPS ;
*
SAUT PAGE;
MESS '                                              ' ;
MESS ' _____________________________________________' ;
MESS '!                                            !' ;
MESS '!    CAS DE DEUX CUBES SUPERPOSES SANS JOINT !' ;
MESS '!                 ENTRE EUX                  !' ;
MESS '!___________________________________________ !' ;
MESS '                                              ' ;
*
OPTION DIME 3 ;
OPTION ELEM QUA4 MODE TRID ;
*
* ----DEFINITION DE LA GEOMETRIE DU CUBE SUPERIEUR-----
*
SA1 = 0.00 0.00 0.00 ;
SB1 = 2.00 0.00 0.00 ;
SC1 = 2.00 2.00 0.00 ;
SD1 = 0.00 2.00 0.00 ;
*
* ----------            MAILLAGE             ----------
*
SH1 =  SA1 DROIT 1 SB1 ;
SH2 =  SB1 DROIT 1 SC1 ;
SH3 =  SC1 DROIT 1 SD1 ;
SH4 =  SD1 DROIT 1 SA1 ;
*
SL1 = SH1 ET SH2 ET SH3 ET SH4 ;
SS1 = SURF SL1 PLAN ;
*
* ----DEFINITION DE LA GEOMETRIE DU CUBE INFERIEUR-----
*
IA1 = 0.00 0.00 -4.00 ;
IB1 = 2.00 0.00 -4.00 ;
IC1 = 2.00 2.00 -4.00 ;
ID1 = 0.00 2.00 -4.00 ;
*
* ----------            MAILLAGE             ----------
*
IH1 =  IA1 DROIT 1 IB1 ;
IH2 =  IB1 DROIT 1 IC1 ;
IH3 =  IC1 DROIT 1 ID1 ;
IH4 =  ID1 DROIT 1 IA1 ;
*
IL1 = IH1 ET IH2 ET IH3 ET IH4 ;
IS1 = SURF IL1 PLAN ;
*
OPTION ELEM CUB8 ;
*
VOLS = SS1 VOLU 1 TRAN ( 0.00 0.00 -2.00 ) ;
FACS = FACE 2 VOLS ;
VOLI = IS1 VOLU 1 TRAN ( 0.00 0.00  2.00 ) ;
FACI = FACE 2 VOLI ;
VT1  = VOLS ET VOLI ;
VT1  = ELIM 0.00001 VT1 ;
VTSAN = VT1 ;
*
* ------- DEFINITION DES CONDITIONS AUX LIMITES -------
*
CL11 = BLOQ IC1 UX ;
CL12 = BLOQ IC1 UY ;
CL13 = BLOQ IC1 UZ ;
CL1  = CL11 ET CL12 ET CL13 ;
*
CL21 = BLOQ IH2 UX ;
CL22 = BLOQ IH2 UZ ;
CL2  = CL21 ET CL22 ;
*
CL31 = BLOQ IH3 UY ;
CL32 = BLOQ IH3 UZ ;
CL3  = CL31 ET CL32 ;
*
CL4  = BLOQ IA1 UZ ;
*
CL   = CL1 ET CL2 ET CL3 ET CL4 ;
*
* ----------      DEFINITION DU MODELE       ----------
*
MOD1 = MODL VT1 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE' ;
MA1  = MATR MOD1 YOUN 2.1E11 NU 0.3 ;
*
* ----------     MATRICE DE RIGIDITE         ----------
*
RI1  = RIGI MOD1 MA1 ;
RI2  = RI1 ET CL ;
*
* ----------       PRESSION UNIFORME         ----------
*
CHPO1= PRES 'MASS' MOD1 -100000.00 SS1 ;
*
* ----------            RESOLUTION           ----------
*
RE   = RESO RI2 CHPO1 ;
*
* ------REDUC. DES DEPLA. AUX DIFFERENTS ELEMENTS------
*
```

```
DEPS = REDU RE VOLS ;
MESS ' DEPLACEMENTS DES NOEUDS DU CUBE SUPERIEUR ' ;
LIST DEPS;
*
desusa=deps;
*
DEPI = REDU RE VOLI ;
MESS ' DEPLACEMENTS DES NOEUDS DU CUBE INFERIEUR ' ;
LIST DEPI ;
*
deinsa=depi;
*
*
* ----------     CALCUL DES DEFORMATIONS     ----------
*
EPS1 = EPSI MOD1 RE ;
*
* ------REDUC. DES DEFOR. AUX DIFFERENTS ELEMENTS------
*
EPSS = REDU EPS1 VOLS ;
MESS ' DEFORMATIONS DU CUBE DU HAUT ' ;
LIST EPSS ;
*
EPSB = REDU EPS1 VOLI ;
MESS ' DEFORMATIONS DU CUBE DU BAS ' ;
LIST EPSB ;
*
* -------        CALCUL DES CONTRAINTES        --------
*
SIG1 = SIGMA MOD1 MA1 RE ;
*
* -----REDUC. DES CONTR. AUX DIFFERENTS ELEMENTS-------
*
SIGS = REDU SIG1 VOLS ;
MESS ' CONTRAINTES DU CUBE DU HAUT ' ;
LIST SIGS ;
*
SIGI = REDU SIG1 VOLI ;
MESS ' CONTRAINTES DU CUBE DU BAS ' ;
LIST SIGI ;
*
MESS '                                              ' ;
MESS '                                              ' ;
MESS' *============================================*';
*
SAUT PAGE;
MESS '                                              ' ;
MESS '                                              ' ;
MESS ' ____________________________________________' ;
MESS '!                                            !' ;
MESS '!  CAS DE DEUX CUBES SUPERPOSES AVEC UN      !' ;
MESS '!          JOINT ENTRE EUX                   !' ;
MESS '!____________________________________________!' ;
MESS '                                              ' ;
MESS '                                              ' ;
*
*
OPTION DIME 3 ;
OPTION ELEM QUA4 MODE TRID ;
*
* -----DEFINITION DE LA GEOMETRIE DU CUBE SUPERIEUR----
*
SA1A = 0.00 0.00 0.00 ;
SB1A = 2.00 0.00 0.00 ;
SC1A = 2.00 2.00 0.00 ;
SD1A = 0.00 2.00 0.00 ;
*
* ----------             MAILLAGE            ----------
*
SH1A =  SA1A DROIT 1 SB1A ;
SH2A =  SB1A DROIT 1 SC1A ;
SH3A =  SC1A DROIT 1 SD1A ;
SH4A =  SD1A DROIT 1 SA1A ;
*
SL1A = SH1A ET SH2A ET SH3A ET SH4A ;
SS1A = SURF SL1A PLAN ;
*
* ----DEFINITION DE LA GEOMETRIE DU CUBE INFERIEUR-----
*
IA1A = 0.00 0.00 -4.00 ;
IB1A = 2.00 0.00 -4.00 ;
IC1A = 2.00 2.00 -4.00 ;
ID1A = 0.00 2.00 -4.00 ;
*
* ----------            MAILLAGE             ----------
*
IH1A =  IA1A DROIT 1 IB1A ;
IH2A =  IB1A DROIT 1 IC1A ;
IH3A =  IC1A DROIT 1 ID1A ;
IH4A =  ID1A DROIT 1 IA1A ;
*
IL1A = IH1A ET IH2A ET IH3A ET IH4A ;
IS1A = SURF IL1A PLAN ;
*
OPTION ELEM CUB8 ;
*
VOLSA = SS1A VOLU 1 TRAN ( 0.00 0.00 -2.00 ) ;
FACSA = FACE 2 VOLSA ;
VOLIA = IS1A VOLU 1 TRAN ( 0.00 0.00  2.00 ) ;
FACIA = FACE 2 VOLIA ;
VT1A  = VOLSA ET VOLIA ;
*
* ------ DEFINITION DES CONDITIONS AUX LIMITES --------
*
CL11A = BLOQ IC1A UX ;
CL12A = BLOQ IC1A UY ;
CL13A = BLOQ IC1A UZ ;
CL1A  = CL11A ET CL12A ET CL13A ;
*
CL21A = BLOQ IH2A UX ;
CL22A = BLOQ IH2A UZ ;
CL2A  = CL21A ET CL22A ;
*
CL31A = BLOQ IH3A UY ;
CL32A = BLOQ IH3A UZ ;
CL3A  = CL31A ET CL32A ;
*
CL4A  = BLOQ IA1A UZ ;
*
CLA   = CL1A ET CL2A ET CL3A ET CL4A ;
*
* ----     DEFINITION DU MODELE DES DEUX CUBES     ----
*
MOD1A = MODL VT1A 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE' ;
MA1A  = MATR MOD1A YOUN 2.1E11 NU 0.3 ;
*
* - CREATION DE L'ELEMENT JOINT ENTRE LES DEUX CUBES  -
*
OPTION ELEM LIA4 ;
VOLJA = LIAISON 0.0001 FACSA FACIA ;
*
VTOTAL=VT1A ET VOLJA ;
*
* ----------  DEFINITION DU MODELE DU JOINT  ----------
*
MODJA = MODL VOLJA 'MECANIQUE' 'ELASTIQUE' 'ISOTROPE'
```

```
       JOI4 ;
MAJA = MATR MODJA KS 4.2E15 KN 4.2E15 ;
*
* ----- MODELE ET MATERIAU DE TOUTE LA STRUCTURE ------
*
MODTA = MOD1A ET MODJA ;
MATTA = MA1A ET MAJA ;
*
* ----------      MATRICE DE RIGIDITE      ----------
*
RITA  = RIGI MODTA MATTA ;
RITOA = RITA ET CLA ;
*
* ----------      PRESSION UNIFORME        ----------
*
CHPO1A= PRES 'MASS' MODTA -100000.00 SS1A ;
*
* ----------            RESOLUTION         ----------
*
RETA  = RESO RITOA CHPO1A ;
*
* ----- REDUC. DES DEPLA. AUX DIFFERENTS ELEMENTS -----
*
DEPSA = REDU RETA VOLSA ;
MESS ' DEPLACEMENTS DES NOEUDS DU CUBE SUPERIEUR ' ;
LIST DEPSA;
*
DEPIA = REDU RETA VOLIA ;
MESS ' DEPLACEMENTS DES NOEUDS DU CUBE INFERIEUR ' ;
LIST DEPIA ;
*
DEPJ = REDU RETA VOLJA ;
MESS ' DEPLACEMENTS DES NOEUDS DU JOINT ' ;
LIST DEPJ;
* ----------   CALCUL DES DEFORMATIONS     ----------
*
EPSTA = EPSI MODTA RETA ;
*
* ---- REDUC. DES DEFOR. AUX DIFFERENTS ELEMENTS ------
*
EPSSA = REDU EPSTA VOLSA ;
MESS ' DEFORMATIONS DU CUBE DU HAUT ' ;
LIST EPSSA ;
*
EPSBA = REDU EPSTA VOLIA ;
MESS ' DEFORMATIONS DU CUBE DU BAS ' ;
LIST EPSBA ;
*
EPSJA = REDU EPSTA VOLJA ;
MESS ' DEFORMATIONS DU JOINT ' ;
LIST EPSJA ;
*
* ----------   CALCUL DES CONTRAINTES      ----------
*
SIGTA = SIGMA MODTA MATTA RETA ;
*
* ----- REDUC. DES CONTR. AUX DIFFERENTS ELEMENTS -----
*
SIGSA = REDU SIGTA VOLSA ;
MESS ' CONTRAINTES DU CUBE DU HAUT ' ;
LIST SIGSA ;
*
SIGIA = REDU SIGTA VOLIA ;
MESS ' CONTRAINTES DU CUBE DU BAS ' ;
LIST SIGIA ;
*
SIGJA = REDU SIGTA VOLJA ;
MESS ' CONTRAINTES DU JOINT ' ;
LIST SIGJA ;

*
* ---------- CODE DE FONCTIONNEMENT ----------
*
*
desuav=deps;
* difference des deplacements configurations avec/sans
* joint pour les cubes superieurs
desdif=desusa-desuav;
*
deinav=depi;
* difference des deplacements configurations avec/sans
* joint pour les cubes inferieurs
deidif=deinsa-deinav;
*
maxdis = maxi desdif 'ABS' ;
maxdii = maxi deidif 'ABS' ;
*
*
si ((maxdis <eg 1e-9) et (maxdii <eg 1e-9));
   erre 0 ;
sino ;
   erre 5 ;
finsi ;
*

*
FIN;
```

Test joi41 **Comments**

1. LIAISON OPERATOR: creation of linkage elements

```
OPTI ELEM LIA4 ;
VOLJA = LIAISON 0.0001 FACSA FACIA ;
```

The LIAISON operator generates the set of linkage elements between two surfacic objects (FACSA and FACIA: MAILLAGE type). VOLJA is the generated object (MAILLAGE type), and 0.0001 is the proximity criterion between the two surfacic objects.

For the creation of a joint element JOI4 (3D):
-FACSA and FACIA represent respectivelly surfaces 1 and 2 of the element. These surfaces should be read in the same direction. This direction is that of the boundary of surface 1. The numbering of the nodes of surface 1 must be such that the axes (1,2,N) make up a direct trihedron, with N orientated positivelly in the direction of the opening of the joint element,
We define the following quantities :
. axis 1 = vector connecting node 1 to node 2 of surface 1
. axis 2 = vector connecting node 1 to node 4 of surface 1
. axis N = vector normal to the plane defined by vectors 1 and 2
. opening of the joint element = separator movement of surface 1
from surface 2 when surface 2 is held fixed

-To take into account initial free space x in a joint element, FACSA and FACIA should have a distance x between them. In addition, x should be input as an initial normal inelastic strain before the use of the NONLIN procedure. (See report DMT/93.655)

LIA4 is a connection element with 2*4 nodes.

2. REDU OPERATOR: restriction of a field or an object

```
DEPS = REDU RE VOLS ; (OBJET3 = REDU  OBJET1  OBJET2 ;)
```

The REDU operator restricts :
- a field by points to the points of a given mesh or to the non-zero values of a CHPOINT
- a field by elements to the elements of a given mesh or to the given MMODEL object
- a MMODEL object to a given mesh
- a NUAGE object to given component

We have different possible types:

OBJET1	*OBJET2*	*OBJET3*
CHPOINT	*MAILLAGE*	*CHPOINT*
CHPOINT	*POINT*	*CHPOINT*
CHPOINT	*CHPOINT*	*CHPOINT*
MCHAML	*MAILLAGE*	*MCHAML*
MCHAML	*MMODEL*	*MCHAML*
MMODEL	*MAILLAGE*	*MMODEL*
NUAGE	*MOT1, MOT2...*	*NUAGE*

In case of restriction of a MCHAML or a MMODEL on a mesh :
- this mesh must be composed of elements of the same type as the mesh that supports the MCHAML or the MMODEL
- all the elements of the mesh must be included in the mesh that supports the MCHAML or the MMODEL

11.3 Test joi44 Description sheet

Test name joi44

Calculation type ORTHOTROPIC ELASTIC MECHANICS **Finite element type** QUA4, JOI4

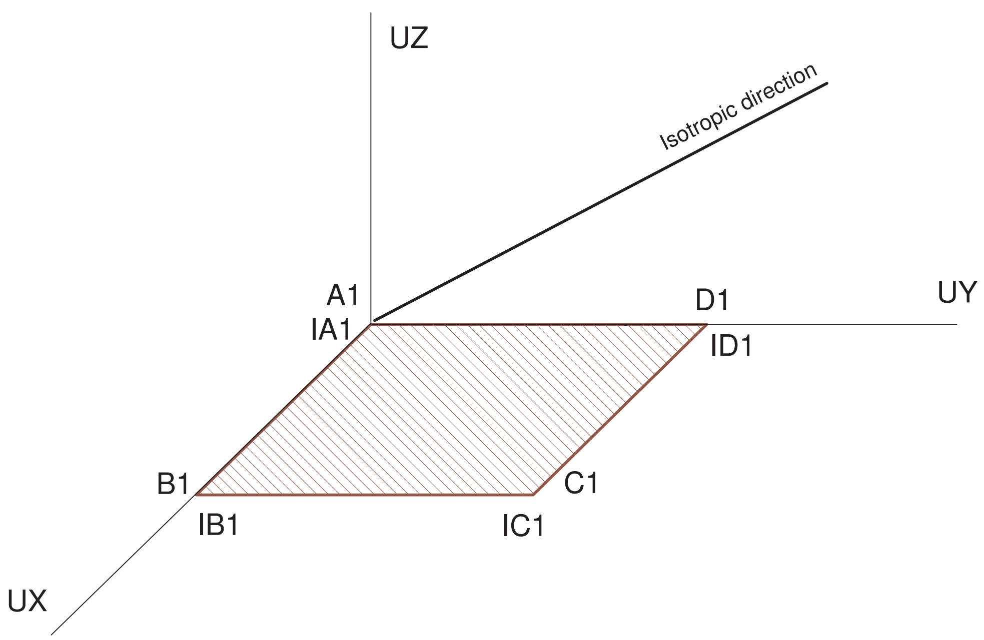
Topic Calculation of the thermical stresses on a 3D joint.

The structure is a parallelepipedic joint embedded at the lower surface. The upper surface is free. The joint is subjected to a field of temperatures of CHAMELEM type, and of subtype TEMPERATURES. The MCHAML is pertaining to a MMODEL object at first, and to a mesh object secondly. **Goal** Calculate the

field of thermial stresses induced by a field of temperatures. **Version** 97' customer version

Model description

KS1	=	$4.2\ 10^{10}$
KS2	=	$3.0\ 10^{5}$
KN	=	$4.2\ 10^{20}$
ALPN	=	2.5
T	=	1000.0



Test joi44 **Results**

RESULTS Analytical solution:

SIGMA = K * ALPN * DELTA(T)
where
K is the stiffness in the direction perpendicular to the joint plan
ALPN is the thermal expansion coefficient in the direction n, perpendicular to the joint plan (ALPN is homogeneous to length per degrees; the length is the thickness of the joint)
DELTA(T) is the CHAMALEM of temperature
Then SIGMA = 4.2E20 * 2.5 * 1000.0
= 1.05E24

Castem solution:

	CHAMELEM ON A MODEL OBJECT		
	SMS1	*SMS2*	*SMSN*
Analytical solution	0.00000E00	0.00000E00	1.05000E24
Castem Solution			
Point 1	0.000E00	0.000E00	1.050E24
Point 2	0.000E00	0.000E00	1.050E24
Point 3	0.000E00	0.000E00	1.050E24
Point 4	0.000E00	0.000E00	1.050E24

	CHAMELEM ON A MESH OBJECT		
	SMS1	*SMS2*	*SMSN*
Analytical solution	0.00000E00	0.00000E00	1.05000E24
Castem Solution			
Point 1	0.000E00	0.000E00	1.050E24
Point 2	0.000E00	0.000E00	1.050E24
Point 3	0.000E00	0.000E00	1.050E24
Point 4	0.000E00	0.000E00	1.050E24

```
*          Test Joi44.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
OPTI ECHO 0 ;
SAUT PAGE ;
TEMPS ;
*===================================================*
*                                                   *
*                       TEST JOI44                  *
*                                                   *
*                 CALCUL DES CONTRAINTES            *
*               THERMIQUES SUR UN JOINT 3D          *
*                                                   *
*     Un joint 3D JOI4 a sa surface inferieure      *
*     encastree. Sa surface superieure est libre.   *
*     Un champ de temperatures est donne, de type   *
*     CHAMELEM, de sous type temperatures avec le   *
*     MCHAML s'appuyant d'une part sur un objet de  *
*     type MAILLAGE, d'autre part sur un objet de   *
*     type MMODEL.                                  *
*     On cherche a calculer le champ des contraintes *
*     thermiques induit par ce champ de temperatures.*
*                                                   *
*     Solution analytique :                         *
*                                                   *
*            SIGMA = K * ALPN * DELTA(T)            *
*     ou                                            *
*     K = raideur dans la direction perpendiculaire  *
*         au plan du joint                          *
*     ALPN = coefficient de dilatation thermique    *
*            suivant la direction n, perpendiculaire *
*            au plan du joint                       *
*            ( ALPN est homogene a une longueur     *
*             divisee par des degres, la longueur   *
*             representant l'epaisseur du joint)    *
*     DELTA(T) = CHAMALEM de temperatures           *
*     SIGMA = 4.2E20 * 2.5 * 1000.0                 *
*           = 1.05E24                               *
*                                                   *
*===================================================*
OPTION DIME 3 ;
OPTION ELEM QUA4 MODE TRID  ;
*
SOLNUL = 0.0     ;
SOLANA = 1.05E24 ;
*
* ------- DEFINITION DE LA SURFACE TOP DU JOINT -------
*
A1 = 0.00 0.00 0.00 ;
B1 = 2.00 0.00 0.00 ;
C1 = 2.00 2.00 0.00 ;
D1 = 0.00 2.00 0.00 ;
*
* ----------          MAILLAGE          ----------
*
O1 = 0.00 0.00 10.00 ;
H1 = A1 DROIT 1 B1 ;
H2 = B1 DROIT 1 C1 ;
H3 = C1 DROIT 1 D1 ;
H4 = D1 DROIT 1 A1 ;
*
L1 = H1 ET H2 ET H3 ET H4 ;
S1 = SURF L1 PLAN ;
*
* ------- DEFINITION DE LA SURFACE BOT DU JOINT -------
*
IA1 = 0.00 0.00 0.00 ;
IB1 = 2.00 0.00 0.00 ;
IC1 = 2.00 2.00 0.00 ;
ID1 = 0.00 2.00 0.00 ;
*
* ----------          MAILLAGE          ----------
*
IH1 = IA1 DROIT 1 IB1 ;
IH2 = IB1 DROIT 1 IC1 ;
IH3 = IC1 DROIT 1 ID1 ;
IH4 = ID1 DROIT 1 IA1 ;
*
IL1 = IH1 ET IH2 ET IH3 ET IH4 ;
IS1 = SURF IL1 PLAN ;
*
* ----------   CREATION DU JOINT JOI4   ----------
*
OPTION ELEM LIA4 ;
VOL = LIAISON 0.00001 S1 IS1 ;
*
* -------- DEFINITION DES CONDITIONS LIMITES ----------
*
CL11 = BLOQ IA1 UX ;
CL12 = BLOQ IA1 UY ;
CL13 = BLOQ IA1 UZ ;
CL1 = CL11 ET CL12 ET CL13 ;
*
CL21 = BLOQ IB1 UX ;
CL22 = BLOQ IB1 UY ;
CL23 = BLOQ IB1 UZ ;
CL2 = CL21 ET CL22 ET CL23 ;
*
CL31 = BLOQ IC1 UX ;
CL32 = BLOQ IC1 UY ;
CL33 = BLOQ IC1 UZ ;
CL3 = CL31 ET CL32 ET CL33 ;
*
CL41 = BLOQ ID1 UX ;
CL42 = BLOQ ID1 UY ;
CL43 = BLOQ ID1 UZ ;
CL4 = CL41 ET CL42 ET CL43 ;
*
CL = CL1 ET CL2 ET CL3 ET CL4 ;
*
* ---------  DEFINITION DU MODELE DU JOINT  ----------
*
P1 = 0.707 0.707 0.707 ;
MOD1 = MODL VOL 'MECANIQUE' 'ELASTIQUE' 'ORTHOTROPE'
       JOI4 ;
MA1  = MATR MOD1 'DIRECTION' P1 'PERPENDICULAIRE'
       KS1 4.2E10 KS2 3.0E5 KN 4.2E20 ALPN 2.5;
*
* ----------      MATRICE DE RIGIDITE       ----------
*
RI1 = RIGI MOD1 MA1 ;
RI2 = RI1 ET CL ;
*
* ----------     CONTRAINTES THERMIQUES     ----------
*
*          CHAMELEM S'APPUYANT SUR UN OBJET MODELE
*
C1T =MANU 'CHML' MOD1 'T' 1000.0 'TYPE' 'TEMPERATURES';
SI1 = THET MOD1 MA1 C1T ;
*
MESS '   CHAMELEM S APPUYANT SUR UN OBJET MODELE    ' ;
MESS '                                              ' ;
MESS ' ____________________________________________ ' ;
MESS '!                                            !' ;
MESS '!           JOINT 3D  JOI4                   !' ;
MESS '!                                            !' ;
```

```
MESS '!  - Surface inferieure du joint encastree.  !' ;
MESS '!  - Surface superieure du joint libre     . !' ;
MESS '!  - Chamelem de temperatures donne DELTA(T) !' ;
MESS '!    s appuyant sur un objet de type modele  !' ;
MESS '!                                            !' ;
MESS '! ==>                                        !' ;
MESS '! Champ des contraintes thermiques = Sigma   !' ;
MESS '!                                            !' ;
MESS '!  Sigma = raideur * coeff. dilat. thermi.   !' ;
MESS '!          * DELTA(T)                        !' ;
MESS '!____________________________________________!' ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS ' Solution Analytique :'                         ;
MESS '                                              ' ;
MESS '    SMS1 =' SOLNUL  ;
MESS '    SMS2 =' SOLNUL  ;
MESS '    SMN  =' SOLANA  ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS ' Solution Calculee :'                           ;
LIST SI1 ;
*
* ----------     CODE DE FONCTIONNEMENT      ----------
*
MAXCO1 = MAXI SI1 'ABS' ;
RESI1 = ABS( (MAXCO1-SOLANA)/SOLANA ) ;
*
*           CHAMELEM S'APPUYANT SUR UN OBJET MAILLAGE
*
C2T = MANU 'CHML' VOL 'T' 1000.0 'TYPE' 'TEMPERATURES';
SI2 = THET MOD1 MA1 C2T ;
MESS '                                              ' ;
MESS '  CHAMELEM S APPUYANT SUR UN OBJET MAILLAGE   ' ;
MESS '                                              ' ;
MESS ' ____________________________________________ ' ;
MESS '!                                            !' ;
MESS '!            JOINT 3D  JOI4                  !' ;
MESS '!                                            !' ;
MESS '! - Surface inferieure du joint encastree.   !' ;
MESS '! - Surface superieure du joint libre     .  !' ;
MESS '! - Chamelem de temperatures donne DELTA(T)  !' ;
MESS '!   s appuyant sur un objet de type maillage !' ;
MESS '!                                            !' ;
MESS '! ==>                                        !' ;
MESS '! Champ des contraintes thermiques = Sigma   !' ;
MESS '!                                            !' ;
MESS '! Sigma = raideur * coeff. dilat. thermi.    !' ;
MESS '!         * DELTA(T)                         !' ;
MESS '!____________________________________________!' ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS ' Solution Analytique :'                         ;
MESS '                                              ' ;
MESS '    SMS1 =' SOLNUL  ;
MESS '    SMS2 =' SOLNUL  ;
MESS '    SMN  =' SOLANA  ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS ' Solution Calculee :'                           ;
MESS '                                              ' ;
LIST SI2 ;
*
MAXCO2 = MAXI SI2 'ABS' ;
RESI2 = ABS( (MAXCO2-SOLANA)/SOLANA ) ;
*
SI ( (RESI1 <EG 1E-4) ET (RESI2 <EG 1E-4) );
   ERRE  0 ;
SINO;
   ERRE  5 ;
FINSI ;
*
FIN ;
```

Test joi44 **Comments**

- MANU OPERATOR: creation of a MCHAML object type

```
C1T = MANU CHML MOD1 T 1000.0 TYPE TEMPERATURES ;
SI1 = THET MOD1 MA1 C1T;
C2T = MANU CHML VOL T 1000.0 TYPE TEMPERATURES ;
SI2 = THET MOD1 MA1 C2T ;
```

The MANU operator enables the user to simply create objects of the following types: MAILLAGE, CHPOINT, SOLUTION, RIGIDITE. Here we have the creation of a MCHAML type object. The MANU CHML operator creates a new field by element (MCHAML type) connected with the VOL object (MAILLAGE type) or with the MOD1 object (MMODEL type). In the first case the MCHAML type object is pertaining to a MMODEL object whereas in the second case, it is pertaining to a mesh object.
The temperature T is the name of a component (MOT type) and 1000.0 is its constant value. The field subtype (MOT type) is TEMPERATURES.
The generated field by element is always expressed at the nodes.

11.4 Test joi45 Description sheet

Test name joi45

Calculation type ORTHOTROPIC ELASTIC MECHANICS **Finite element type** QUA4, JOI4

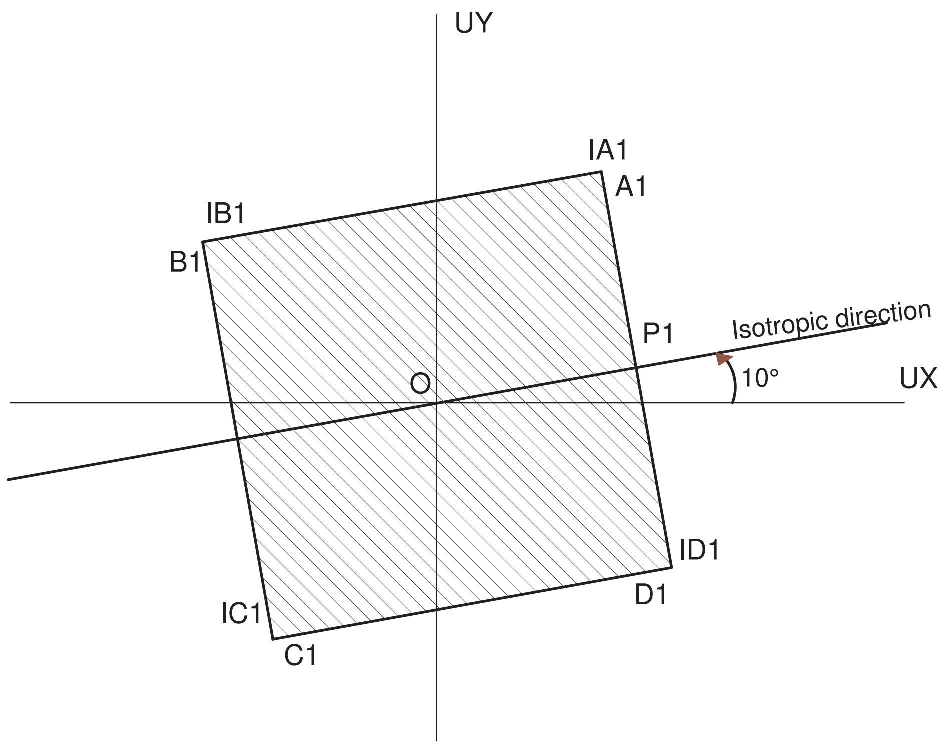
Topic Calculation of the thermal stresses on a 3D joint.

The structure is a parallelepipedic joint embedded from the lower surface. The upper surface is free. The joint is subjected to a shear load applied on the whole circumference of the upper surface in the direction O-P1. **Goal** Calculate the displacements in the direction of the load, the deformations, the stresses and the nodal force field resulting from a stress field integration. **Version** 97' customer version

Model description

$$\begin{aligned} KS1 &= 4.2\ 10^{10} \\ KS2 &= 4.2\ 10^{8} \\ KN &= 4.2\ 10^{20} \end{aligned}$$

Angle (OX,OP1) = 55 degrees
O-P1 axis : first orthotropic direction



Test joi45 **Results**

RESULTS Analytical solution:

$$\text{delta} = \frac{F}{K * A}$$

where

F is the total load on the upper surface in the direction OP1

K is the stiffness in the first orthotropic direction

A is the surface of the joint

Projection of the solution in the axis X and Y:

deltax = delta * cos(55)

deltay = delta * sin(55)

Numerical application:

F = 100000

K = 4.2 E20

A = 4.0

deltax = 3.416 E-7

deltay = 4.875 E-7

```
*           Test Joi45.dgibi: Jeux de données         *
*           --------------------------------          *
*                                                     *
OPTI ECHO 0 ;
SAUT PAGE ;
*=====================================================*
*                                                     *
*                       TEST JOI45                    *
*                                                     *
*        ESSAI DE CISAILLEMENT SUR UN JOINT 3         *
*                     ORTHOTROPE                      *
*                                                     *
*      Un joint 3D JOI4 a sa surface inferieure       *
*      encastree. Sa surface superieure est libre.    *
*      Un effort de cisaillement est exerce sur le    *
*      pourtour de sa surface superieure, suivant la  *
*      direction O-P1.                                *
*                                                     *
*                    Y  !                             *
*                      .!.                            *
*                    .  !  .                          *
*                  .    !    . P1                     *
*                .      !    / .                      *
*              .        !   /    .                    *
*            .          !  /       .                  *
*          .          O ! /          .                *
*      ------------------------------------- X        *
*          .            !            .                *
*            .          !          .                  *
*              .        !        .                    *
*                .      !      .                      *
*                  .    !    .                        *
*                    .  !  .                          *
*                      .!.                            *
*                       !                             *
*                                                     *
*      Angle (OX,OP1) = 55 degres                     *
*      L axe OP1 definit la premiere direction        *
*      d orthotropie                                  *
*                                                     *
*      Calcul de la solution analytique :             *
*                                                     *
*                           F                         *
*             delta = -------------                   *
*                         K  *  A                     *
*      ou                                             *
*                                                     *
*     delta = deplacement suivant la direction        *
*             de la force                             *
*         F = Force totale exercee sur la surface     *
*             superieure, suivant la direction OP1    *
*         K = raideur dans la premiere direction      *
*             d orthotropie                           *
*         A = surface du joint                        *
*                                                     *
* Projection de la solution suivant les axes X et Y : *
*                                                     *
*          deltax = delta * cos(55)                   *
*                                                     *
*          deltay = delta * sin(55)                   *
*=====================================================*
OPTION DIME 3 ;
OPTION ELEM QUA4 MODE TRID  ;
*
OO = 00.00 0.00 0.00 ;
ON = 00.00 0.00 1.00 ;
AN = 10.00 ;
*
```

```
* -------- DEFINITION DU POINT P1 --------------
* ----- POUR LES DIRECTIONS D'ORTHOTROPIE ------
*
P1 = 0.707106781 0.707106781 0.00 ;
P1 = P1 TOUR AN OO ON ;
*
* ------ DEFINITION DE LA SURFACE BOT DU JOINT --------
*
A1 = 1.414213562   0.000000000   0.00 ;
B1 = 0.000000000   1.414213562   0.00 ;
C1 =-1.414213562   0.000000000   0.00 ;
D1 = 0.000000000  -1.414213562   0.00 ;
*
A1 = A1 TOUR AN OO ON ;
B1 = B1 TOUR AN OO ON ;
C1 = C1 TOUR AN OO ON ;
D1 = D1 TOUR AN OO ON ;
*
* ----------          MAILLAGE          ----------
*
H1 = A1 DROIT 1 B1 ;
H2 = B1 DROIT 1 C1 ;
H3 = C1 DROIT 1 D1 ;
H4 = D1 DROIT 1 A1 ;
*
L1 = H1 ET H2 ET H3 ET H4 ;
S1 = SURF L1 PLAN ;
*
* ------- DEFINITION DE LA SURFACE TOP DU JOINT -------
*
IA1 = 1.414213562   0.000000000   0.00 ;
IB1 = 0.000000000   1.414213562   0.00 ;
IC1 =-1.414213562   0.000000000   0.00 ;
ID1 = 0.000000000  -1.414213562   0.00 ;
*
IA1 = IA1 TOUR AN OO ON ;
IB1 = IB1 TOUR AN OO ON ;
IC1 = IC1 TOUR AN OO ON ;
ID1 = ID1 TOUR AN OO ON ;
*
* ----------          MAILLAGE          ----------
*
IH1 = IA1 DROIT 1 IB1 ;
IH2 = IB1 DROIT 1 IC1 ;
IH3 = IC1 DROIT 1 ID1 ;
IH4 = ID1 DROIT 1 IA1 ;
*
IL1 = IH1 ET IH2 ET IH3 ET IH4 ;
IS1 = SURF IL1 PLAN ;
*
* ----------   CREATION DU JOINT JOI4   ----------
*
OPTION ELEM LIA4 ;
VOL = LIAISON 0.00001 S1 IS1 ;
*
* -------- DEFINITION DES CONDITIONS LIMITES ----------
*
CL11 = BLOQ A1 UX ;
CL12 = BLOQ A1 UY ;
CL13 = BLOQ A1 UZ ;
CL1 = CL11 ET CL12 ET CL13 ;
*
CL21 = BLOQ B1 UX ;
CL22 = BLOQ B1 UY ;
CL23 = BLOQ B1 UZ ;
CL2 = CL21 ET CL22 ET CL23 ;
*
CL31 = BLOQ C1 UX ;
```

```
CL32 = BLOQ C1 UY ;
CL33 = BLOQ C1 UZ ;
CL3 = CL31 ET CL32 ET CL33 ;
*
CL41 = BLOQ D1 UX ;
CL42 = BLOQ D1 UY ;
CL43 = BLOQ D1 UZ ;
CL4 = CL41 ET CL42 ET CL43 ;
*
CL = CL1 ET CL2 ET CL3 ET CL4 ;
*
* -------    DEFINITION DU MODELE DU JOINT    ---------
*
MOD1 = MODL VOL 'MECANIQUE' 'ELASTIQUE' 'ORTHOTROPE'
       JOI4 ;
MA1 = MATR MOD1 'DIRECTION' P1 'PARALLELE' KS1 4.2E10
                                           KS2 4.2E08
                                           KN  4.2E20 ;
*********************************************
FLOT1 = 0.0 ;
P3 = 3.0 1.0 1.0 ;
***MA1 = MATR MOD1 'DIRECTION' P1 'INCLINE' FLOT1 P3
***      KS1 4.2E10 KS2 4.2E08 KN  4.2E20 ;
*********************************************
**MA1 = MATR MOD1 'RADIAL' P1 KS1 4.2E10 KS2 4.2E08 KN
*                                            4.2E20 ;
*
* ----------       MATRICE DE RIGIDITE       ---------
*
RI1 = RIGI MOD1 MA1 ;
RI2 = RI1 ET CL ;
*
* ----------        FORCE DE TRACTION        ---------
*
FO1 = FORCE ( 57358.00  81915.00  0.00 ) IL1 ;
*
* ----------            RESOLUTION           ---------
*
RE = RESO RI2 FO1 ;
SAUT PAGE ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS '                                              ' ;
MESS'!=============================================!';
MESS'!                                             !';
MESS'!              TEST JOI45                     !';
MESS'!                                             !';
MESS'!     ESSAI DE CISAILLEMENT SUR UN JOINT 3D   !';
MESS'!                ORTHOTROPE                   !';
MESS'!                                             !';
MESS'!  Un joint 3D JOI4 a sa surface inferieure   !';
MESS'!  encastree. Sa surface superieure est libre. !';
MESS'!  Un effort de cisaillement est exerce sur le !';
MESS'!  pourtour de sa surface superieure, suivant  !';
MESS'!  la direction O-P1.                          !';
MESS'!                                             !';
MESS'!                Y  !                         !';
MESS'!                  .!.                        !';
MESS'!                 . !  .                      !';
MESS'!               .   !    . P1                 !';
MESS'!             .     !    /  .                 !';
MESS'!           .       !   /     .               !';
MESS'!        .          !  /        .             !';
MESS'!      .         O  ! /           .           !';
MESS'!   ----------------------------------- X     !';
MESS'!       .           !           .             !';
MESS'!         .         !         .               !';
MESS'!           .       !       .                 !';
```

```
MESS'!             .     !     .                   !';
MESS'!               .   !   .                     !';
MESS'!                 . !  .                      !';
MESS'!                  .!.                        !';
MESS'!                   !                         !';
MESS'!                                             !';
MESS'!  Angle (OX,OP1) = 55 degres                 !';
MESS'!  L axe OP1 definit la premiere direction    !';
MESS'!  d orthotropie                              !';
MESS'!                                             !';
MESS'!                                             !';
MESS'!  Calcul de la solution analytique :         !';
MESS'!                                             !';
MESS'!                     F                       !';
MESS'!        delta = -------------                !';
MESS'!                  K  *  A                    !';
MESS'!  ou                                         !';
MESS'!                                             !';
MESS'! delta = deplacement suivant la direction    !';
MESS'!        de la force                          !';
MESS'!     F = Force totale exercee sur la surface !';
MESS'!        superieure,suivant la direction OP1 !';
MESS'!     K = raideur dans la premiere direction  !';
MESS'!        d orthotropie                        !';
MESS'!     A = surface du joint                    !';
MESS'!                                             !';
MESS'! Projection de la solution suivant           !';
MESS'! les axes X et Y :                           !';
MESS'!                                             !';
MESS'!        deltax = delta * cos(55)             !';
MESS'!                                             !';
MESS'!        deltay = delta * sin(55)             !';
MESS'!                                             !';
MESS'! Application numerique :                     !';
MESS'!                                             !';
MESS'!     F = 100000                              !';
MESS'!     K = 4.2E20                              !';
MESS'!     A = 4.0                                 !';
MESS'!                                             !';
MESS'!  deltax = 3.416E-7                          !';
MESS'!  deltay = 4.875E-7                          !';
MESS'!                                             !';
MESS'!=============================================!';
MESS '                                              ' ;
MESS '                                              ' ;
SAUT PAGE ;
MESS ' Solution Calculee :'                           ;
MESS '                                              ' ;
*
SOLAX=3.416E-7 ;
SOLAY=4.875E-7 ;
*
LIST RE ;
*
* ----------    CODE DE FONCTIONNEMENT      ----------
*
DEPIAX = EXTR RE UX IA1 ;
RESI1 = ABS( (DEPIAX-SOLAX)/SOLAX ) ;
*
DEPIAY = EXTR RE UY IA1 ;
RESI2 = ABS( (DEPIAY-SOLAY)/SOLAY ) ;
*
SI ( (RESI1 <EG 1E-3 ) ET (RESI2 <EG 1E-3 ) ) ;
   ERRE  0 ;
SINO;
   ERRE  5 ;
FINSI ;
*
```

```
* ----------    CALCUL DES DEFORMATIONS      ----------
*
EPS1 = EPSI MOD1 MA1 RE ;
LIST EPS1 ;
*
* ----------    CALCUL DES CONTRAINTES       ----------
*
SIG1 = SIGMA MOD1 MA1 RE ;
LIST SIG1 ;
*
* ----------        CALCUL DE BSIGMA         ----------
*
BSI1 = BSIGMA MOD1 SIG1 ;
LIST BSI1 ;
*
FIN ;
```

Test joi45 **Comments**

- BSIGMA OPERATOR: creation of a MCHAML object type

```
BSI1 = BSIGMA MOD1 SIG1 ;
```

The BSIGMA operator calculates the nodal force field (BST1) resulting from a stress field (SIG1) integration.

Chapter 12

DYNAMICS

12.1 Test dyna6 Description sheet

Test name dyna6

Calculation type MECHANICS DYNAMICS 3D

Finite element type POUT

Topic
Bending mode shape of a clamped-free beam.

The beam is clamped at one end and free on the other end. The problem consists in finding frequencies of the first and second mode shapes of the beam.

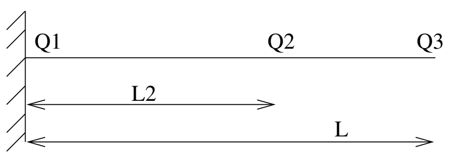
The three options of the VIBR operator are tested in this example :
- within intervals (keyword INTERVALLE) : separation of mode shapes by dichotomy on a given interval
- by proximity (keyword PROCHE) : computation of mode shapes which are close to the given values
- by Lanczos method (keyword SIMULANE) : projection on a subspace

Goal Find the frequencies of the first and second mode shapes of the beam.

Reference Analytical solution : Timoshenko.

Version 97' customer version

Model description



$L = 2m$
$L2 = 0.783 * L$
$A = 0.01m^2$
$I_y = 6.25.10^{-6}m^4$
$I_z = 1.10^{-3}m^4$
$G = 1.10^{-2}m^4$

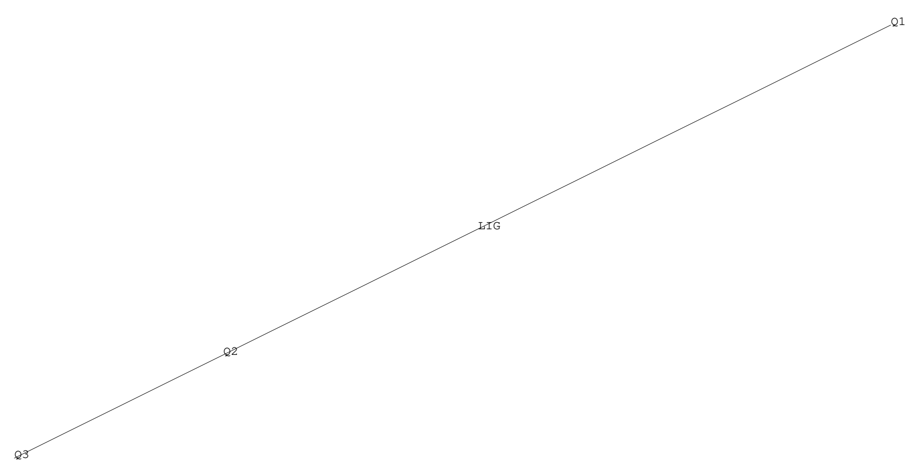
E $= 2.10^{11}Pa$
$\nu = 0.3$
$\rho = 7800kg/m^3$

Test dyna6 **Results**

Results

	Analytic	INTERVALLE	PROCHE	SIMULTANE
Freq (Hz) Shape1	17.71	17.704	17.704	17.704
Freq (Hz) Shape2	110.98	110.71	110.71	110.71
$U_z(Q2)(m)$ Shape2	0.0	$1.2480.10^{-3}$	$1.2483.10^{-3}$	$1.2483.10^{-3}$

CASTEM figures



BENDING OF A CLAMPED-FREE BEAM

```
*           Test Dyna6.dgibi: Jeux de données        *
*           ---------------------------------        *
*                                                    *
OPTI ECHO 0;
GRAPH='N';
SAUT PAGE ;
*
******************************************************
*                          DYNA6
*
*             CAS TEST OPERATEUR VIBRATION
*             MODES DE FLEXION D'UNE POUTRE
*                   ENCASTREE-LIBRE
*
* Cet exemple permet de tester les trois options
* de l'opérateur VIBR :
*
*    - par intervalle (INTERVALLE) o on sépare les
*      modes par dichotomie sur un intervalle donné
*    - par proximité (PROCHE) o on recherche les
*      modes proches de valeurs données
*    - par la méthode de Lanczos (SIMULTANE) o on
*      projette sur un sous-espace
*
*
* Le problème consiste à trouver les fréquences pour
* le premier et le deuxième mode, d'une poutre
* encastrée à une extrémité et libre à l'autre.
*
* Les valeurs obtenues par les trois méthodes sont
* comparées aux valeurs théoriques.
*
******************************************************
*
TITRE 'FLEXION D UNE POUTRE ENCASTREE-LIBRE';
OPTIO DIME 3 ELEM SEG2 MODE TRID IMPI 0;
TEMPS ;
DENSITE 0.1;
*
*---------------------- MAILLAGE ----------------------
*----------- Q2 EST UN NOEUD POUR LE MODE NUMERO 2 ----
*
Q1 = 0. 0. 0.;Q2 = (2.*0.783) 0. 0. ;Q3 = 2. 0. 0. ;
LIG= Q1 D Q2 D Q3 ;
OEIL1 = 5 5 5 ;

SI(NEG GRAPH 'N');
   TRAC OEIL1 LIG ;
FINSI;
*
STAN= MODL LIG MECANIQUE POUT ;

*
*------------ MATERIAU ET CARATERISTIQUES -------------
*
MATPL1=MATR STAN YOUNG 2.E11 NU 0.3 RHO 7800;
CARPL1=CARB STAN SECT 0.01 INRY 6.25E-6 INRZ 1.E-3
      TORS 1.E-2;
MATPL1=MATPL1 ET CARPL1;
*
*------------------ RIGIDITE ET MASSE -----------------
*
RIGPL1 =RIGI STAN MATPL1;
MASPLA1=MASS STAN MATPL1 ;
*
*------------------ ON BLOQUE Q1 ----------------------
*
BLOQ1=BLOQ DEPL ROTA Q1 ;
* RIGPLA1=RIGPL1 ET BLOQ1 ;
* *
* *-----------------------------------------------------
*           PREMIERE OPTION  (INTERVALLE)
*           ON SEPARE LES MODES PAR DICHOTOMIE
*           PUIS ON LANCE LES ITERATIONS INVERSES
*-----------------------------------------------------
*
MODPLA1=VIBR INTE 10. 120. RIGPLA1 MASPLA1 ;
*
*-----------------------------------------------------
*           DEUXIEME OPTION  (PROCHE)
*           ON LANCE LES ITERATIONS INVERSES
*           A PARTIR DES FREQUENCES CONTENUES
*           DANS LE LISTREEL
*-----------------------------------------------------
*
PR=PROG 17. 110.;
MODPLA2=VIBR PROC PR RIGPLA1 MASPLA1 ;
*
*-----------------------------------------------------
*           TROISIEME OPTION  (SIMULTANE)
*           ON UTILISE LA METHODE DE LANCZOS
*           (PROJECTION SUR UN SOUS-ESPACE)
*-----------------------------------------------------
*
MODPLA3=VIBR SIMU 1. 2 RIGPLA1 MASPLA1 ;
*
*---------------------- R{sultats ---------------------
*
OPTI ECHO 0;
SAUTER PAGE;
F1=TIRE MODPLA1 FREQ RANG 1;
G1=TIRE MODPLA1 FREQ RANG 2;
F2=TIRE MODPLA2 FREQ RANG 1;
G2=TIRE MODPLA2 FREQ RANG 2;
F3=TIRE MODPLA3 FREQ RANG 1;
G3=TIRE MODPLA3 FREQ RANG 2;
*
MESS '*';
MESS '********    ON VERIFIE LES FREQUENCES';
MESS '*';
MESS
'          ANALYTIQUE   *   INTERVALLE    *     PROCHE
      *    SIMULTANE';
MESS
'       -------------*-----------------*-----------
-----*-------------';
MESS ' MODE 1    17.71      *   ' F1 '*' F2 '*    ' F3;
MESS ' MODE 2   110.98      *   ' G1 '*' G2 '*    ' G3;
MESS ' ';
MESS ' ';
MESS '*';
MESS
'********    ON VERIFIE LA DEFORMEE DU DEUXIEME MODE';
MESS '*';
MESS ' ';
X1=EXTR (TIRE MODPLA1 DEPL RANG 2) UZ Q2;
X2=EXTR (TIRE MODPLA2 DEPL RANG 2) UZ Q2;
X3=EXTR (TIRE MODPLA3 DEPL RANG 2) UZ Q2;
MESS
'         ANALYTIQUE   *   INTERVALLE    *     PROCHE
      *    SIMULTANE';
MESS
'         -------------*-----------------*-----------
-----*-------------';
MESS 'MODE 2     0.00      *  ' X1 '*' X2 '*   ' X3;
TEMPS ;
```

```
*          CODE FONCTIONNEMENT
FREF=17.71;  GREF=110.98;
RESI1=ABS((F1-FREF)/FREF); RESI2=ABS((G1-GREF)/GREF);
SI((RESI1 <EG 1E-2) ET (RESI2 <EG 1E-2));
  ERRE 0;
SINO;
  ERRE 5;
FINSI;
FIN;
```

Test dyna6 **Comments**

1. **Computation of rigidity and mass matrices**

   RIGPL1 = RIGI STAN MATPL1;
   MASPLA1= MASS STAN MATPL1 ;

   The operators RIGI and MASS require the model (here STAN) and the material and geometrical characteristics (here MATPL1).
   The type of these two objects is RIGIDITE. The subtype of RIGPL1 is RIGIDITE and the subtype of MASPLA1 is MASSE.

2. **Computation of eigen shapes : option INTERVALLE**

   MODPLA1 = VIBR INTE 10. 120. RIGPLA1 MASPLA1 ;

   The operator VIBR requires the rigidity and mass matrices of the system. VIBR with keyword INTE computes the natural eigen shapes which frequencies are contained between 10 Hz and 120 Hz.

3. **Computation of eigen shapes : option PROCHE**

   PR = PROG 17. 110.;
   MODPLA2 = VIBR PROC PR RIGPLA1 MASPLA1 ;

   VIBR with keyword PROC computes the natural eigen shapes which frequencies are the nearest to the given list (here PR).

4. **Computation of eigen shapes : option SIMULTANE**

   MODPLA3 = VIBR SIMU 1. 2 RIGPLA1 MASPLA1 ;

   VIBR with keyword SIMU computes the two natural eigen shapes which frequencies are the nearest to the given real (here 1.).

12.2 Test dyna7 Description sheet

Test name dyna7

Calculation type MECHANICS DYNAMICS 2D FOURIER

Finite element type LQUA4, LSU2

Topic
Vibrating mode shapes of a thin fluid web.

The cavity is supposed undeformable. The gravity effect is taken into consideration.
The shaking mode shapes are computed in an annular space with Fourier modes number 1 and 3.
The three options of the VIBR operator are tested in this example.

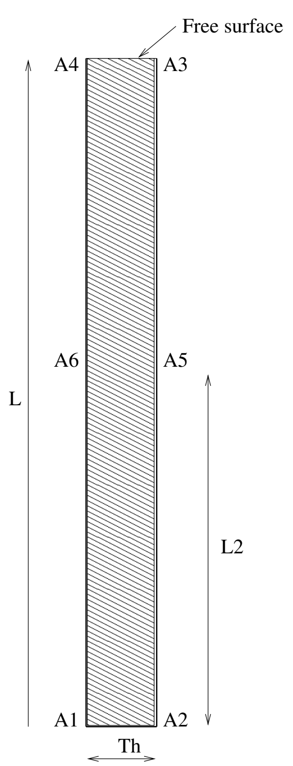
Goal Find the frequencies of the first and third Fourier shape.

Reference Analytical solution.

Version 97' customer version

Model description

$L = 2m$	$\rho = 1000kg/m^3$	$\rho_{ref} = 1kg/m^3$
$L2 = 1.2m$	$S_s = 1200m/s$	$S_{ref} = 1m/s$
$Th = 0.01m$	$G = 9.81m/s^2$	$L_{car} = 1m$



Test dyna7 **Results**

Results

	Analytical	INTERVALLE	PROCHE	SIMULTANE
Freq (Hz) Shape1	0.489	0.4895	0.4895	0.4895
Freq (Hz) Shape3	0.863	0.8638	0.8638	0.8638
P(A1) Shape1	1.	1.	1.	1.
P(A2) Shape1	1.81	1.8108	1.8108	1.8108
P(A3) Shape1	3.76	3.763	3.763	3.763
P(A1) Shape3	1.	1.	1.	1.
P(A2) Shape3	18.31	18.375	18.375	18.375
P(A3) Shape3	201.72	202.86	202.86	202.86

CASTEM figures

SHAKING SHAPES OF A THIN FLUID WEB

```
*           Test Dyna7.dgibi: Jeux de données        *
*           ---------------------------------        *
*                                                    *
OPTI ECHO 0;
GRAPH='N';
SAUT PAGE ;
*
******************************************************
*                          DYNA7
*               CAS TEST OPERATEUR VIBRATION
*
*
* Cet exemple permet de tester les trois options de
* l'opérateur VIBR
*
* On calcule les modes de ballotement d'une lame mince
* de fluide dans un espace annulaire en mode de
* Fourier 1 et 3.
*
* On suppose la cavité indéformable. L'effet de la
* gravité est pris en compte.
*
*
******************************************************
TITRE 'MODES DE BALLOTEMENT D UNE LAME MINCE DE FLUIDE';
OPTI DIME 2 ELEM QUA4;
GRAPH='N';
TEMPS ;

*--------------------------- MAILLAGE -----------------
*----------------- ON NE MAILLE QUE LE FLUIDE ---------
*
A1=0.995 0.;A2=1.005 0.;A3=1.005 2.;A4=0.995 2.;
A5=1.005 1.2;A6=0.995 1.2;
L1=D 1 A1 A2;
L2=A2 D 24 A5 D 16 A3;
L3=D 1 A3 A4;
L4=A4 D 16 A6 D 24 A1;
FLUID=DALL L1 L2 L3 L4;
*
SI (NEG GRAPH 'N');
TRAC FLUID ;
FINSI;
*
OPTI ELEM SEG2;
SURFA=D 1 A3 A4;
*
SI (NEG GRAPH 'N');
TRAC SURFA ;
FINSI;
*
MAIL=FLUID ET SURFA;
*
SI (NEG GRAPH 'N');
TRAC MAIL;
FINSI;
*
*---------- DEFINITION DE QUELQUES CONSTANTES ---------
*
ROREF=1.;CEREF=1.;LCARA=1.;
RO0=1000.;CS=1200.;GRA=9.81;
*
*-- MODELE ET OBJET AFFECTE FLUIDE ET SURFACE LIBRE ---
*-------- POUR L'HARMONIQUE DE FOURIER 1 -------------
*
OPTI MODE FOUR 1;
MODLIQ1=MODL FLUID LIQUIDE LQU4;
MODLIQ2=MODL SURFA LIQUIDE LSU2;
MODLIQ=MODLIQ1 ET MODLIQ2;
*
*--------------- MATERIAU LIQUIDE ---------------------
*
MAT=MATR MODLIQ RHO RO0 RORF ROREF CSON CS CREF CEREF
    LCAR LCARA G GRA;
*
*--------------- RIGIDITE ET MASSE --------------------
*
RAI=RIGI MODLIQ MAT;
MAS=MASS MODLIQ MAT;
*
*_____________________________________________________
* ON CALCULE LE PREMIER MODE AVEC L'OPTION INTERVALLE
* DE VIBR. ON RENORMALISE LE CHAMP DE DEPLACEMENT
* (ET DE PRESSION) DE MANIERE A CE QUE LA PRESSION SOIT
* EGALE A 1 A L'ALTITUDE 0 ET ON EXTRAIT LES PRESSIONS
* AUX POINTS A2, A5 ET A3
*_____________________________________________________
*
MOD1=VIBR INTE 0. 1. RAI MAS;
F1=TIRE MOD1 FREQ RANG 1;
PRE=TIRE MOD1 DEPL RANG 1;
PBAS=EXTR PRE P A2;
PP=PRE/PBAS;
X1=EXTR PP P A2;Y1=EXTR PP P A5;Z1=EXTR PP P A3;
*
*_____________________________________________________
* ON CALCULE LE PREMIER MODE AVEC L'OPTION
* PROCHE DE VIBR
*_____________________________________________________
*
MOD2=VIBR PROC (PROG 0.5) RAI MAS;
F2=TIRE MOD2 FREQ RANG 1;
PRE=TIRE MOD2 DEPL RANG 1;
PBAS=EXTR PRE P A2;
PP=PRE/PBAS;
X2=EXTR PP P A2;Y2=EXTR PP P A5;Z2=EXTR PP P A3;
*
*_____________________________________________________
* ON CALCULE LE PREMIER MODE AVEC L'OPTION SIMULTANE
* DE VIBR
*_____________________________________________________
*
MOD3=VIBR SIMU 0.5 1 RAI MAS;
F3=TIRE MOD3 FREQ RANG 1;
PRE=TIRE MOD2 DEPL RANG 1;
PBAS=EXTR PRE P A2;
PP=PRE/PBAS;
X3=EXTR PP P A2;Y3=EXTR PP P A5;Z3=EXTR PP P A3;
*
*_____________________________________________________
* ON RECOMMENCE LES OPERATIONS CETTE FOIS
* AVEC L'HARMONIQUE 3
*_____________________________________________________
*
OPTI MODE FOUR 3;
MODLIQ1=MODL FLUID LIQUIDE LQU4;
MODLIQ2=MODL SURFA LIQUIDE LSU2;
*
MODLIQ=MODLIQ1 Et MODLIQ2;
*
MAT=MATR MODLIQ RHO RO0 RORF ROREF CSON CS CREF CEREF
    LCAR LCARA G GRA;
*
RAI=RIGI MODLIQ MAT;
MAS=MASS MODLIQ MAT;
*
```

```
MOD1=VIBR INTE 0. 1. RAI MAS;
*--------------------------------------------------------
G1=TIRE MOD1 FREQ RANG 1;
PRE=TIRE MOD1 DEPL RANG 1;
PBAS=EXTR PRE P A2;
PP=PRE/PBAS;
T1=EXTR PP P A2;U1=EXTR PP P A5;V1=EXTR PP P A3;
*--------------------------------------------------------
MOD2=VIBR PROC (PROG 0.9) RAI MAS;
*
G2=TIRE MOD2 FREQ RANG 1;
PRE=TIRE MOD2 DEPL RANG 1;
PBAS=EXTR PRE P A2;
PP=PRE/PBAS;
T2=EXTR PP P A2;U2=EXTR PP P A5;V2=EXTR PP P A3;
*--------------------------------------------------------
MOD3=VIBR SIMU 0.9 1 RAI MAS;
G3=TIRE MOD3 FREQ RANG 1;
PRE=TIRE MOD3 DEPL RANG 1;
PBAS=EXTR PRE P A2;
PP=PRE/PBAS;
T3=EXTR PP P A2;U3=EXTR PP P A5;V3=EXTR PP P A3;
*--------------------------------------------------------
SAUT PAGE; SAUT 2 LIGNE ;
MESS '*';
MESS '********    ON VERIFIE LES FREQUENCES';
MESS '*';
MESS ' ';
MESS
'         ANALYTIQUE   *   INTERVALLE    *     PROCHE
     *    SIMULTANE';
MESS
'         -------------*-----------------*-----------
-----*-------------';
MESS ' HARM 1    0.489      *   ' F1 '*' F2 '*    ' F3;
MESS ' HARM 3    0.863      *   ' G1 '*' G2 '*    ' G3;
MESS ' ';
MESS '*';
MESS
'********   ON VERIFIE LE CHAMP DE PRESSION AUX POINTS
A1,A5 ET A3';
MESS '*';
MESS ' ';
MESS
'        ANALYTIQUE   *   INTERVALLE    *     PROCHE
     *    SIMULTANE';
MESS
'        -------------*-----------------*-----------
-----*-------------';
MESS 'HARM 1     1.00      *  ' X1 '*' X2 '*   ' X3 ;
MESS 'HARM 1     1.81      *  ' Y1 '*' Y2 '*   ' Y3 ;
MESS 'HARM 1     3.76      *  ' Z1 '*' Z2 '*   ' Z3 ;
MESS 'HARM 3     1.00      *  ' T1 '*' T2 '*   ' T3 ;
MESS 'HARM 3    18.31      *  ' U1 '*' U2 '*   ' U3 ;
MESS 'HARM 3   201.72      *  ' V1 '*' V2 '*   ' V3 ;
TEMPS ;
*------------- code de bon fonctionnement ------------*
SAUT 1 PAGE;
ERR=TABLE;
ERR.1=100*(ABS(0.489 -F1)/0.489);
ERR.2=100*(ABS(0.489-F2)/0.489);
ERR.3=100*(ABS(0.489 -F3)/0.489);
ERR.4=100*(ABS(0.863 -G1)/0.863);
ERR.5=100*(ABS(0.863 -G2)/0.863);
ERR.6=100*(ABS(0.863 -G3)/0.863);
ERR.7=100*(ABS(1-X1)/1);
ERR.8=100*(ABS(1-X2)/1);
ERR.9=100*(ABS(1-X3)/1);
ERR.10=100*(ABS(1.81 -Y1)/1.81);
ERR.11=100*(ABS(1.81 -Y2)/1.81);
ERR.12=100*(ABS(1.81 -Y3)/1.81);
ERR.13=100*(ABS(3.76 -Z1)/3.76);
ERR.14=100*(ABS(3.76 -Z2)/3.76);
ERR.15=100*(ABS(3.76 -Z3)/3.76);
ERR.16=100*(ABS(1-T1));
ERR.17=100*(ABS(1-T2));
ERR.18=100*(ABS(1-T3));
ERR.19=100*(ABS(18.31 -U1)/18.31);
ERR.20=100*(ABS(18.31 -U2)/18.31);
ERR.21=100*(ABS(18.31 -U3)/18.31);
ERR.22=100*(ABS(201.72-V1)/201.72);
ERR.23=100*(ABS(201.72-V2)/201.72);
ERR.24=100*(ABS(201.72-V3)/201.72);

I=0;
REPETER BOUC1 24;
I=I+1;
SI (ERR.I <EG 5);
 SAUT 1 LIGN;
 ERRE 0;
SINON;
 ERRE 5;
 MESS 'erreur valeur ' I ' du tableau';
FINSI;
FIN BOUC1;
*-----------------------------------------------------*
FIN;
```

Test dyna7 **Comments**

1. **Liquid formulation**

   MODLIQ1 = MODE FLUID LIQUIDE LQU4;
   MODLIQ2 = MODE SURFA LIQUIDE LSU2;

   The operator MODEL with keyword LIQUIDE defines a liquid formulation. The type of finite elements depends of the type of formulation. LQU4 is a four node massive quadrangle and LSU2 is a two node element of free surface.

2. **Liquid material**

   MAT=MATE MODLIQ RHO RO0 RORF ROREF CSON CS CREF CEREF LCAR LCARA G GRA;

   The material characteristics are defined with MATER operator and use the following parameters :

   - RHO : volumic mass
   - RORF : referential volumic mass
   - CSON : sonic speed
   - CREF : referential speed
   - LCAR : characteristic length
   - G : gravitational acceleration

3. **Results extraction**

   PRE = TIRE MOD1 DEPL RANG 1;
   PBAS = EXTR PRE P A2;

   The object PRE is a ponctual field and one of its components is the pressure P. The data PBAS is the pressure at point A2.

12.3 Test fsi4 Description sheet

Test name fsi4

Calculation type MECHANICS DYNAMICS 2D FOURIER
FLUID-STRUCTURE INTERACTION

Finite element type COQ2, LQUA4, RACO

Topic
Frequencies of concentric shells coupled by fluid.

Both the concentric shells are separated by a volume of water. Springs are on the bottom of the shells. The inner radius is 0.5m and the outer radius is 1m.

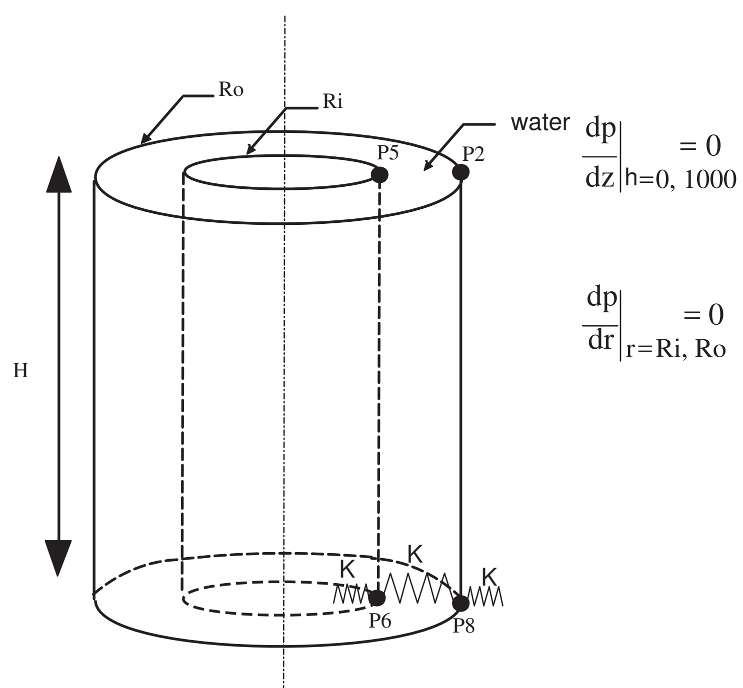
Goal Find the frequencies of the first and second mode shapes of the shells (M=1).

Reference CASTEM Report CEA 91/479 : Validation of CASTEM 2000 for fluid structure interaction problems.

Version 97' customer version

Model description

R_i	$= 500mm$	E_{shell}	$= 2.10^5 MPa$	G	$= 0mm/s^2$
R_o	$= 1000mm$	ν_{shell}	$= 0.3$	ρ_{ref}	$= 1000kg/m^3$
H	$= 1000mm$	ρ_{water}	$= 1000kg/m^3$	S_{ref}	$= 1435.10^3 mm/s$
Th_{shell}	$= 5mm$	S_s	$= 1435.10^3 mm/s$	L_{car}	$= 1000mm$
ρ_{shell}	$= 8000kg/m^3$				

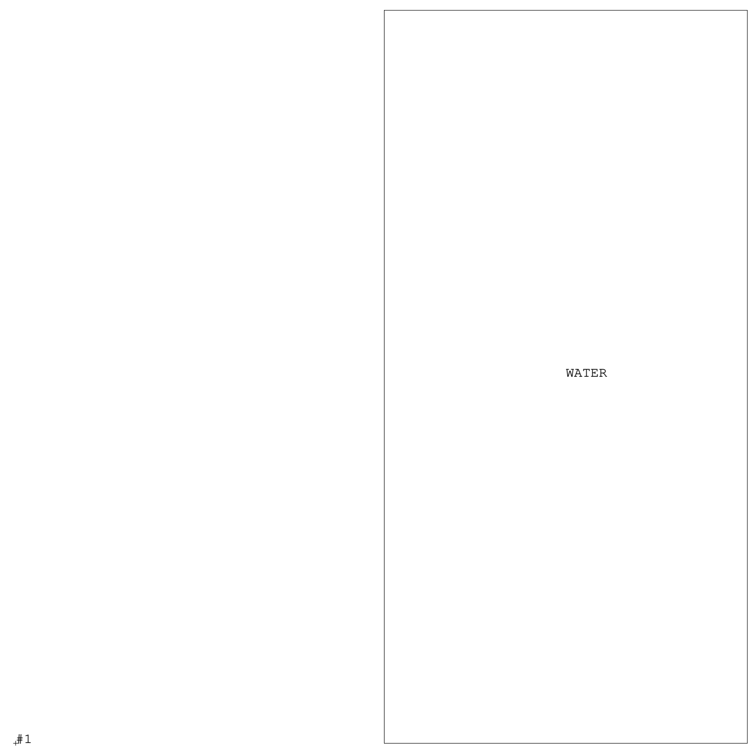


Test fsi4 **Results**

Results

	Theoretical freq (Hz)	CASTEM freq (Hz)	Margin
Shape1	18.88	18.647	1.24 %
Shape2	55.278	56.083	1.46 %

CASTEM figures



FSI4 : MESHING

```
*          Test Fsi4.dgibi: Jeux de données
*          --------------------------------
*
* CAS TEST DU 91/10/04     PROVENANCE : PETI
OPTI ECHO 0;
SAUT PAGE;
*
*****************************************************
*                                                   *
*                      TEST FSI4                    *
*                                                   *
*          CONCENTRIC SHELLS COUPLED BY FLUID       *
*                                                   *
*  Calculation of the frequencies of two concentric *
*  shells with springs coulped by water in between. *
*  The inner radius is 0.5m and the outer radius is *
*  1.0m                                             *
*                                                   *
*                                                   *
*               P5         P7                       *
*                |         |                        *
*                |         |                        *
*                |         |                        *
*                |  fluid  |                        *
*                |         |                        *
*                |         |                        *
*                |         |                        *
*                |         |                        *
*         ^^^^^^^ ^^^^^^^^^ ^^^^^^^                 *
*           K   P6    K   P8   K                    *
*                                                   *
*  The boundary conditions are                      *
*                                                   *
*      dp  |                                        *
*     ---- |                = 0.                    *
*      dz  | h = 0. , 1.0                           *
*                                                   *
*      dp  |                                        *
*     ---- |                = 0.                    *
*      dr  | r = Ri , Ro                            *
*****************************************************
*
TEMPS;
*
OPTI DIME 2;
OPTI MODE FOUR 1  ;
OPTI ELEM QUA4;
*
GRAPH = 'N';
*
*====================================================*
*               geometry                             *
*====================================================*
*
* Dimensions en metres
*
* Points
*
N1 = 1        ; N2 = 1           ;
P5 = 500.0 1000.0 ; P6 = 500.0 0.0 ;
P7 = 1000. 1000.0 ; P8 = 1000. 0.0 ;
*
*
*
S5 =P5 D N1 P6 ; S6 = P8 D N1 P7 ;
*
STEEL = S5 ET S6                 ;
WATER = (S5 PLUS (2.5 0.))  TRAN N2 (495.0 0. )    ;
```

```
RAC1  = RACC 3.0   WATER STEEL   ;
*
* OPTIO FOR TRACE
*
SI (NEG GRAPH 'N');
  TITR ' FSI4 : MAILLAGE';
  TRAC QUAL (WATER ET (0 0));
FINSI;
*
*====================================================*
*    modele - affecte - materiau - rigidite - masse  *
*====================================================*
*
MODCOQ=MODL STEEL MECANIQUE COQ2 ;
MODLIQ=MODL WATER LIQUIDE  LQU4;
MODRAC=MODL RAC1 MECANIQUE LIQUIDE RACO;

*
MATCOQ=MATR MODCOQ RHO 8.E-9  YOUN 2.E5  NU 0.3 ;
MATLIQ = MATR (MODLIQ ET MODRAC)
         RHO 1.E-9  RORF 1.E-9  CSON 1435.0E3 CREF
         1435.0E3 LCAR 1000. G 0.;
CARCOQ=CARB MODCOQ EPAI 5.0  ;
CARRAC=CARB MODRAC LIQU WATER;
MATCOQ=MATCOQ ET CARCOQ;
MATLIQ=MATLIQ ET CARRAC;
*
K    =3.7458E4/2.0;
PP   =PROG K (-1*K) K  ;
PPNEW = PROG K ( -1*K) ( -1*K) K;
RIG1 = RIGI  (MODCOQ ET MODLIQ ET MODRAC)
             (MATCOQ ET MATLIQ)   ;
MAS4 = MASS  (MODRAC ET MODLIQ  )(MATLIQ)  ;
*
*
*====================================================*
*               boundary conditions                  *
*====================================================*
*
RES1 = BLOQ UZ RT  (S5 ET S6)       ;
RIG2 = (APPU UR UT K P8 ) ET (APPU UR UT K P6 ) ;
RIG3 = ( MANU 'RIGI' type 'RIGIDITE' (P6 D 1 P8)
       (MOTS UR)  PPNEW )
    ET ( MANU 'RIGI' type 'RIGIDITE' (P6 D 1 P8)
       (MOTS UT)  PPNEW );
RIG4 = RIG1 ET RIG2        ET RIG3 ;              ;
*
*====================================================*
* theritical value  of frequency
*
FRE1  = TABLE ;
FRE1.1 = 18.88 ;
FRE1.2 = 55.278;
LIST1 = PROG FRE1.1 FRE1.2 ;            ;
*
*====================================================*
*             calculation of the frequencies         *
*                          and                       *
*               extraction of some results           *
*====================================================*
*
*
* Use of the operator VIBR. (option PROC)
*
*
RESUL = VIBR PROC LIST1       (RES1 ET RIG4 )  MAS4  ;
*
*====================================================*
```

```
*                   results                          *
*====================================================*
*
MESS ' RESULTATS ';
MESS ' --------- ';
SAUT 1 LIGN;
*
*
FRE2 = TABLE; MOD = TABLE; DEF = TABLE;ERG = TABLE;
*
I = 0;
REPETER BLOC1 2;
  I = I + 1;
  FRE2.I = TIRE RESUL FREQ RANG I;
  ERG.I = 100 * (ABS ((FRE1.I - FRE2.I) / FRE1.I));
  MESS ' MODE '  I     ;
  MESS ' ----------';
  MESS ' Frequence theorique :' FRE1.I 'Hz';
  MESS ' Frequence calculee  :' FRE2.I 'Hz';
  MESS '    Soit un ecart de : ' ERG.I '%';
  SAUT 1 LIGN;
FIN BLOC1;
*
*====================================================*
*                   code validation                  *
*====================================================*
*
ERGMAX = MAXI (PROG ERG.1 ERG.2 );
*
SI (ERGMAX <EG 5.);
   ERRE 0;
SINON;
   ERRE 5;
FINSI;
*
SAUT 1 LIGN;
TEMPS;
SAUT 1 LIGN;
*
FIN;
*
```

Test fsi4 **Comments**

1. **Union of fluid and structure**

   WATER = (S5 PLUS (2.5 0.)) TRAN N2 (495.0 0.) ;
   RAC1 = RACC 3.0 WATER STEEL ;

   If the distance between points of STEEL and WATER is smaller than 3.0 mm, the operator RACCORD creates a line of double points by joining the STEEL structure with the WATER fluid. This corresponds to the creation of union elements between STEEL and WATER.
   Note that there is a clearance between WATER boundaries and STEEL median line, corresponding here with the thickness of the shell.

2. **Declaration of models**

   MODCOQ=MODEL STEEL MECANIQUE COQ2 ;
   MODLIQ=MODEL WATER LIQUIDE LQU4;
   MODRAC=MODEL RAC1 MECANIQUE LIQUIDE RACO;

   Fluid-structure interaction requires three different models : the liquid, the solid and the model of union elements.

3. **Declaration of material and geometrical characteristics**

   MATLIQ = MATER (MODLIQ ET MODRAC) RHO 1.E-9
   RORF 1.E-9 CSON 1435.0E3 CREF 1435.0E3
   LCAR 1000. G 0.;
   ...
   CARRAC=CARAC MODRAC LIQU WATER;

   The definition of material properties is the same for the liquid model and the union model.
   The liquid model has no geometrical characteristics but the position of the liquid has to be known : hence the operator CARAC is used with the keyword LIQU to indicate that the liquid concerning the union elements of MODRAC is represented by the object WATER.

4. **Modelling of springs**

   RIG2 = (APPU UR UT K P8) ET (APPU UR UT K P6) ;
   RIG3 = (MANU 'RIGI' type 'RIGIDITE' (P6 D 1 P8) (MOTS UR) PPNEW)
   ET (MANU 'RIGI' type 'RIGIDITE' (P6 D 1 P8) (MOTS UT) PPNEW);

   Springs can be modelled by two methods :
   - with operator APPUI if the spring is connected to a fixed point,
   - with operator MANUEL RIGI if the spring joins two points of the structure. In this case, a stiffness is added to each point.

12.4 Test dyna5 Description sheet

Test name dyna5

Calculation type MECHANICS DYNAMICS 3D
SUB-STRUCTURATION

Finite element type POUT

Topic
Methods of sub-structuration.

The problem consists in using sub-structuration methods to find the natural frequency of a clamped-clamped beam. The beam is decomposed in two pieces of same length.

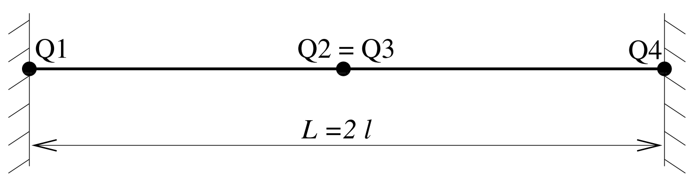
The three methods correspond to the three different ways to describe the constraints between the two common tips Q2 and Q3 :
- Q2 free - Q3 free
- Q2 clamped - Q3 clamped
- Q2 clamped - Q3 free

Goal Find the first natural frequency of the beam.

Reference Analytical solution.

Version 97' customer version

Model description



l	$= 2m$		
A	$= 0.1m^2$	E	$= 2.10^{11}Pa$
I_y	$= 2.08.10^{-7}m^4$	ν	$= 0.3$
I_z	$= 3.33.10^{-8}m^4$	ρ	$= 7800kg/m^3$
I_O	$= 2.41.10^{-7}m^4$		

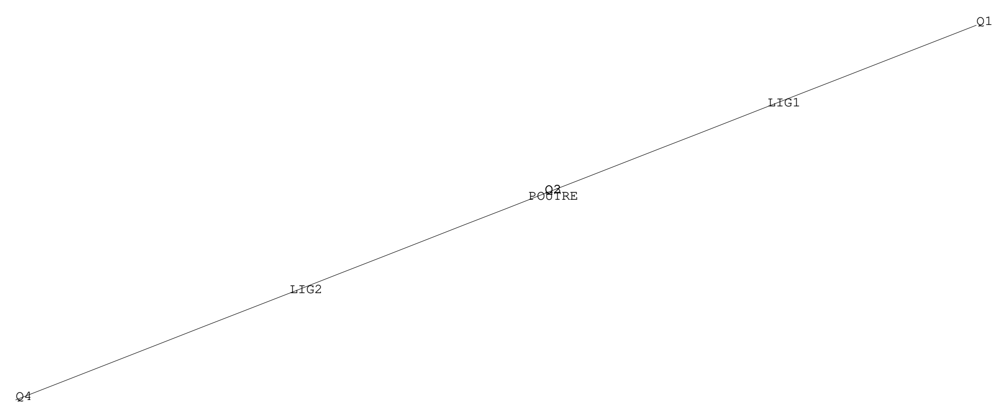
Test dyna5

Results

Results

	Analytic	1st method	2nd method	3rd method
Freq (Hz)	0.65	0.65072	0.65083	0.64926

CASTEM figures



DYNA5 - METHODS OF SUB-STRUCTURATION

```
*          Test Dyna5.dgibi: Jeux de données
*          ---------------------------------
*
* CAS TEST DU 92/01/07     PROVENANCE : PHIL
OPTI ECHO 0;
GRAPH='N';
SAUT PAGE ;
******************************************************
*
* DYNA5 - Test de sous-structuration par 3 méthodes
*
* La fréquence propre d'une poutre est égale à 65 Hz.
* On se propose de retrouver cette fréquence par trois
* méthodes de sous-structuration.
*
* Pour cela on décompose la poutre en deux morceaux
* de meme longueur.
*
*
*           Q1 poutre 1   Q2
*           .--------------..------------.
*                           Q3 poutre 2  Q4
*
*
* Les trois méthodes de sous-structuration correspon-
* dent aux conditions aux limites de l'extrémité
* commune Q2 et Q3 des deux morceaux de poutres :
*              * Q2 libre  - Q3 libre
*              * Q2 bloqu{ - Q3 bloqué
*              * Q2 bloqu{ - Q3 libre
*
* A ces trois méthodes correspondent trois manières de
* décrire les liaisons entre sous-structures
* (opérateur RELA).
*
******************************************************
*
TITRE 'DYNA5 - METHODES DE SOUS-STRUCTURATION' ;
OPTIO DIME 3 ELEM SEG2 MODE TRID IMPI 0;
OPTION  TRAC BENS;
SOL=TABLE;
TEMPS ;
*
*_____________________________________________________
*   PREMIERE METHODE LIBRE-LIBRE
*_____________________________________________________
*
*---- Définition de la géométrie de la 1ère poutre ----
*
Q1 = 0. 0. 0.; Q2 =2. 0. 0.;
LIG1= D 12 Q1 Q2;
OEIL1 = 5 5 5 ;
*                TRACE OEIL1 LIG1 ;
*
*---- Type de calcul, matériau et caractéristiques ----
*---------------- de la 1ère poutre -------------------
*
MO1= MODL LIG1 MECANIQUE ELASTIQUE POUT ;

MATPL1=MATR MO1  YOUNG 2.E11 NU 0.3 RHO 7800;
CARPL1=CARB MO1 SECT 0.1 INRY 2.08E-7 INRZ 3.33E-8
TORS 2.41E-7;
MATPL1=MATPL1 ET CARPL1;
*
*------ Rigidité, masse et conditions aux limites -----
*----------------- de la 1ère poutre ------------------
*
RIGPL1 =RIGI MATPL1 MO1 ;
* MASPLA1=MASS MATPL1 MO1 ;
* BLOQ1=BLOQ DEPL ROTA Q1 ;
* RIGPLA1=RIGPL1 ET BLOQ1 ;
*
*---- Recherche des modes propres de la 1ère poutre ---
*
PR=PROG 0.4 ;
MODPLA1=VIBR PROC PR RIGPLA1 MASPLA1;
*
*---- Définition de la géométrie de la 2ème poutre ----
*
Q3 =2. 0. 0.; Q4 =4. 0. 0.;
LIG2= D 12 Q3 Q4;
*                TRAC OEIL1 LIG2 ;
*
POUTRE=LIG1 ET LIG2 ;
SI(NEG GRAPH 'N');
    TRAC OEIL1 POUTRE;
FINSI;
*
*---- Type de calcul, matériau et caractéristiques ----
*---------------- de la 2ème poutre -------------------
*
MO2= MODL LIG2 MECANIQUE ELASTIQUE POUT ;
MATPL2=MATR MO2 YOUNG 2.E11 NU 0.3 RHO 7800;
CARPL2=CARB MO2 SECT 0.1 INRY 2.08E-7 INRZ 3.33E-8
TORS 2.41E-7;
MATPL2=MATPL2 ET CARPL2;
*
*----- Rigidité, masse et conditions aux limites ------
*---------------- de la 2ème poutre -------------------
*
RIGPL2 =RIGI MATPL2 MO2 ;
MASPLA2=MASS MATPL2 MO2 ;
BLOQ4=BLOQ DEPL ROTA Q4;
RIGPLA2=RIGPL2 ET BLOQ4 ;
*
*---- Recherche des modes propres de la 2ème poutre ---
*
MODPLA2=VIBR PROC PR RIGPLA2 MASPLA2;
*
* Création des sous-structures associées aux 2 poutres
*
SPLA1=STRU RIGPLA1 MASPLA1 ;
SPLA2=STRU RIGPLA2 MASPLA2 ;
ELM2=ELST Q2 SPLA1;
ELM3=ELST Q3 SPLA2;
PR2=PROG 1. ;PR3=PROG -1. ;
*
*--- Description des liaisons entre sous-structures ---
*
LIUX=RELA ELM2 UX PR2 ELM3 UX PR3;
LIUY=RELA ELM2 UY PR2 ELM3 UY PR3;
LIUZ=RELA ELM2 UZ PR2 ELM3 UZ PR3;
LIRX=RELA ELM2 RX PR2 ELM3 RX PR3;
LIRY=RELA ELM2 RY PR2 ELM3 RY PR3;
LIRZ=RELA ELM2 RZ PR2 ELM3 RZ PR3;
ENCL=LIUY ET LIRX ET LIRY ET LIRZ ET LIUX ET LIUZ;
*
* -Création des bases modales associées aux 2 poutres -
*
BASE1=BASE SPLA1 MODPLA1 ENCL ;
BASE2=BASE SPLA2 MODPLA2 ENCL ;
*
*------------------ Base modale totale ----------------
*---------------- correspondant à la réunion des ------
*------------------- deux bases élémentaires ----------
*
```

```
BASET= BASE1 ET BASE2 ;
RIGMO  = RIGI BASET ;
MASMO  = MASS BASET ;
*
*------------ Recherche de la 1ère fréquence ----------
*------------ propre de la structure totale -----------
*
SOL.1= VIBR PROC (PROG 0.65) MASMO RIGMO ;
FREQ1=TIRE SOL.1 FREQ RANG 1;
*
*
*______________________________________________________
*   DEUXIEME METHODE BLOQUE-BLOQUE
*______________________________________________________
*
*
*-------------------- Conditions aux limites ----------
*--------------- sur l'extrémité de la 1ère poutre ----
*
BLOQUX2=BLOQ UX Q2;
BLOQUY2=BLOQ UY Q2;
BLOQUZ2=BLOQ UZ Q2;
BLOQRX2=BLOQ RX Q2;
BLOQRY2=BLOQ RY Q2;
BLOQRZ2=BLOQ RZ Q2;
*
*------------ Rigidité de la 1ère poutre --------------
*
RIGPLA1=RIGPL1 ET BLOQ1
ET BLOQUX2 ET BLOQUY2 ET BLOQUZ2 ET BLOQRX2 ET
BLOQRY2 ET BLOQRZ2;
*
*---- Recherche des modes propres de la 1ère poutre ---
*
PR1=PROG 2.6 ;
MODPLA1=VIBR PROC PR1 RIGPLA1 MASPLA1;
*
*---------------- Conditions aux limites --------------
*------------ sur l'extrémité de la 2ème poutre -------
*
*- Le point Q2 de la 1ère poutre et le point Q3 de la -
*-------- 2ème poutre ont les memes coordonnées -------
*
BLOQUX3=BLOQ UX Q3;
BLOQUY3=BLOQ UY Q3;
BLOQUZ3=BLOQ UZ Q3;
BLOQRX3=BLOQ RX Q3;
BLOQRY3=BLOQ RY Q3;
BLOQRZ3=BLOQ RZ Q3;
*
*------------ Rigidité de la 2ème poutre --------------
*
RIGPLA2=RIGPL2 ET BLOQ4
ET BLOQUX3 ET BLOQUY3 ET BLOQUZ3 ET BLOQRX3 ET
BLOQRY3 ET BLOQRZ3;
*
*---- Recherche des modes propres de la 2ème poutre ---
*
MODPLA2=VIBR PROC PR1 RIGPLA2 MASPLA2;
*
* Création des sous structures associées aux 2 poutres
*
SPLA1=STRU RIGPLA1 MASPLA1 ;
SPLA2=STRU RIGPLA2 MASPLA2 ;
CLUX2=CLST SPLA1 BLOQUX2;
CLUY2=CLST SPLA1 BLOQUY2;
CLUZ2=CLST SPLA1 BLOQUZ2;
CLRX2=CLST SPLA1 BLOQRX2;
CLRY2=CLST SPLA1 BLOQRY2;
CLRZ2=CLST SPLA1 BLOQRZ2;
CLUX3=CLST SPLA2 BLOQUX3;
CLUY3=CLST SPLA2 BLOQUY3;
CLUZ3=CLST SPLA2 BLOQUZ3;
CLRX3=CLST SPLA2 BLOQRX3;
CLRY3=CLST SPLA2 BLOQRY3;
CLRZ3=CLST SPLA2 BLOQRZ3;
*
*--- Description des liaisons entre sous-structures ---
*
PR2=PROG 1. 1.;PR3=PROG 1. 1.;
LIUX=RELA CLUX2 LX PR2 CLUX3 LX PR3;
LIUY=RELA CLUY2 LX PR2 CLUY3 LX PR3;
LIUZ=RELA CLUZ2 LX PR2 CLUZ3 LX PR3;
LIRX=RELA CLRX2 LX PR2 CLRX3 LX PR3;
LIRY=RELA CLRY2 LX PR2 CLRY3 LX PR3;
LIRZ=RELA CLRZ2 LX PR2 CLRZ3 LX PR3;
ENCL=LIUY ET LIRX ET LIRY ET LIRZ ET LIUX ET LIUZ;
*
*--------- Création des bases modales élémentaires ----
*------------- associées aux deux poutres ------------
*
BASE1=BASE SPLA1 MODPLA1 ENCL ;
BASE2=BASE SPLA2 MODPLA2 ENCL ;
*
*------------------ Base modale totale ----------------
*------------- correspondant à la réunion des ---------
*---------------- deux bases élémentaires -------------
*
BASET= BASE1 ET BASE2 ;
RIGMO  = RIGI BASET ;
MASMO  = MASS BASET ;
*
*------------- Recherche de la 1ère fréquence ---------
*-------------- propre de la structure totale ---------
*
SOL.2= VIBR PROC (PROG 0.65) MASMO RIGMO ;
FREQ2=TIRE SOL.2 FREQ RANG 1;
*
*
*______________________________________________________
*   TROISIEME METHODE LIBRE-BLOQUE
*______________________________________________________
*
*
*------------ Rigidité et conditions aux limites ------
*----------------- de la 1ère poutre ------------------
*
RIGPLA1=RIGPL1 ET BLOQ1
ET BLOQUX2 ET BLOQUY2 ET BLOQUZ2 ET BLOQRX2 ET BLOQRY2
ET BLOQRZ2;
*
*---- Recherche des modes propres de la 1ère poutre ---
*
PR1=PROG 2.6;
MODPLA1=VIBR PROC PR1 RIGPLA1 MASPLA1;
*
*-------- Rigidité et conditions aux limites ----------
*--------------- de la 2ème poutre --------------------
*
RIGPLA2=RIGPL2 ET BLOQ4;
*
*---- Recherche des modes propres de la 2ème poutre ---
*
PR2=PROG 0.41 ;
MODPLA2=VIBR PROC PR2 RIGPLA2 MASPLA2;
*
```

```
*------------ Définition des sous-structures ----------
*
SPLA2=STRU RIGPLA2 MASPLA2 ;
CL3=ELST SPLA2 Q3;
*
*---- Définition des liaisons entre sous-structures ---
*
PR=PROG 1 1;
REL1=RELA CL3 UX 1 CLUX2 LX PR ;
REL2=RELA CL3 UY 1 CLUY2 LX PR ;
REL3=RELA CL3 UZ 1 CLUZ2 LX PR ;
REL4=RELA CL3 RX 1 CLRX2 LX PR ;
REL5=RELA CL3 RY 1 CLRY2 LX PR ;
REL6=RELA CL3 RZ 1 CLRZ2 LX PR ;
ENCL=REL1 ET REL2 ET REL3 ET REL4 ET REL5 ET REL6;
*
*--------- Création des bases modales élèmentaires ----
*
BASE1=BASE SPLA1 MODPLA1 ENCL ;
BASE2=BASE SPLA2 MODPLA2 ENCL ;
*
*------------------ Base modale totale ----------------
*------------- correspondant à la réunion des ---------
*----------------- deux bases élémentaires ------------
*
BASET= BASE1 ET BASE2 ;
RIGMO  = RIGI BASET ;
MASMO  = MASS BASET ;
*
*---------- Recherche de la 1ère fréquence ------------
*---------- propre de la structure totale -------------
*
SOL.3= VIBR PROC (PROG 0.65) MASMO RIGMO ;
FREQ3=TIRE SOL.3 FREQ RANG 1;
*_____________________________________________________
*   RESULTATS
*_____________________________________________________
SAUT PAGE ; SAUT 4 LIGNE ;
MESS 'ON DOIT TROUVER UNE FREQUENCE DE 0.65 HZ';
SAUT 2 LIGNE ;
MESS 'PAR LA PREMIERE  METHODE ON TROUVE : ' FREQ1;
MESS 'PAR LA DEUXIEME  METHODE ON TROUVE : ' FREQ2;
MESS 'PAR LA TROISIEME METHODE ON TROUVE : ' FREQ3;
TEMPS ;
*          CODE FONCTIONNEMENT
REF=0.65;
RESI1=ABS((FREQ1-REF)/REF);
RESI2=ABS((FREQ2-REF)/REF);
RESI3=ABS((FREQ3-REF)/REF);
SI((RESI1 <EG 5E-2) ET (RESI2 <EG 5E-2) ET
  (RESI3 <EG 5E-2));
    ERRE  0;
SINO;
    ERRE  5;
FINSI;
FIN;
```

Test dyna5 **Comments**

1. **Creation of sub-structures related to the two beams**

```
SPLA1=STRU RIGPLA1 MASPLA1 ;
SPLA2=STRU RIGPLA2 MASPLA2 ;
ELM2=ELST Q2 SPLA1;
ELM3=ELST Q3 SPLA2;
...
CLUX2=CLST SPLA1 BLOQUX2;
CLUY2=CLST SPLA1 BLOQUY2;
CLUZ2=CLST SPLA1 BLOQUZ2;
CLRX2=CLST SPLA1 BLOQRX2;
CLRY2=CLST SPLA1 BLOQRY2;
CLRZ2=CLST SPLA1 BLOQRZ2;
```

The operator STRU creates an object of STRUCTURE type by indicating the mass and the rigidity.
The operator ELST creates an object of ELEMSTRU type by indicating the structure and the point of the interface. This type of object is used to describe the constraints between sub-structures when the point of the interface is free.
The operator CLST creates an object of BLOQSTRU type by indicating the structure and a rigidity of fixing. This type of object is used to describe the constraints between sub-structures when the point of the interface is constrained.

2. **Description of elementary constraints**

• Interface with free points

```
PR2=PROG 1. ; PR3=PROG -1. ;
LIUX=RELA ELM2 UX PR2 ELM3 UX PR3;
LIUY=RELA ELM2 UY PR2 ELM3 UY PR3;
LIUZ=RELA ELM2 UZ PR2 ELM3 UZ PR3;
LIRX=RELA ELM2 RX PR2 ELM3 RX PR3;
LIRY=RELA ELM2 RY PR2 ELM3 RY PR3;
LIRZ=RELA ELM2 RZ PR2 ELM3 RZ PR3;
```

The operator RELA describes elementary constraints between the sub-structures, it creates an object of ATTACHE type.
The element of structure ELM2 (that is to say Q2 point) has the same displacements as the element ELM3 (Q3 point).

• Interface with fixed point

```
PR2=PROG 1. 1.;PR3=PROG 1. 1.;
LIUX=RELA CLUX2 LX PR2 CLUX3 LX PR3;
LIUY=RELA CLUY2 LX PR2 CLUY3 LX PR3;
LIUZ=RELA CLUZ2 LX PR2 CLUZ3 LX PR3;
LIRX=RELA CLRX2 LX PR2 CLRX3 LX PR3;
LIRY=RELA CLRY2 LX PR2 CLRY3 LX PR3;
LIRZ=RELA CLRZ2 LX PR2 CLRZ3 LX PR3;
```

The fixing of structure CLUX2 (related to Q2 X-displacement) has the same component as CLUX3.
Note that 'LX' is the compulsory component for a fixing.

- Interface with fixed and free points

  CL3=ELST SPLA2 Q3;
  ...
  PR=PROG 1 1;
  REL1=RELA CL3 UX 1 CLUX2 LX PR ;
  REL2=RELA CL3 UY 1 CLUY2 LX PR ;
  REL3=RELA CL3 UZ 1 CLUZ2 LX PR ;
  REL4=RELA CL3 RX 1 CLRX2 LX PR ;
  REL5=RELA CL3 RY 1 CLRY2 LX PR ;
  REL6=RELA CL3 RZ 1 CLRZ2 LX PR ;

  It is possible to describe a constraint between a fixed point and a free point, in this case the operator RELA is used with an element of structure (CL3) and a fixing of structure (CLUX2, CLUY2, ...).

3. **Creation of modal basis**

   BASE1=BASE SPLA1 MODPLA1 ENCL ;
   BASE2=BASE SPLA2 MODPLA2 ENCL ;

   The operator BASE creates an object of BASEMODA type. This object contains :
   - the structure (SPLA1),
   - the mode shapes related to this structure (MODPLA1),
   - the constraints (ENCL).

4. **Reunion of the elementary bases**

   BASET = BASE1 ET BASE2 ;
   RIGMO = RIGI BASET ;
   MASMO = MASS BASET ;

   The two modal bases are joined together.
   The operators RIGI and MASS calculate rigidity and mass matrices by using a modal basis (BASET).

12.5 Test vibr6 Description sheet

Test name vibr6

Calculation type MECHANICS DYNAMICS 3D
COMPOUND EIGEN SHAPES

Finite element type TUYA

Topic
Compound eigen modes of a rod.

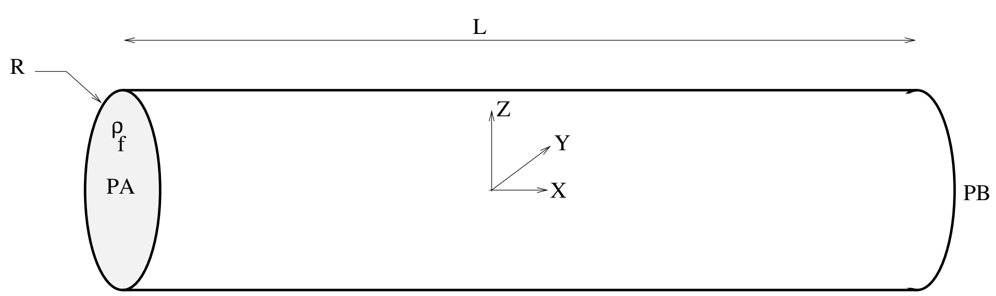
The rod is subjected to an external axial permanent flow. Both the tips of the rod are articulated. Computations are made for different flow velocities (0 up to 120 m/s).

Goal Find the real and imaginary parts of frequencies for the different velocities.

Reference Model of Paidoussis & Ortoja-Starzewski.

Version 97' customer version

Model description



L	$= 2m$		
R	$= 10^{-3}m$	ρ_{fl}	$= 1000kg/m^3$
Th	$= 10^{-3}m$	V_{min}	$= 0m/s$
ρ	$= 4000kg/m^3$	V_{max}	$= 120m/s$
E	$= 10^{10}Pa$	$step_{velo}$	$= 1m/s$
ν	$= 0$		

Test vibr6 **Results**

Results

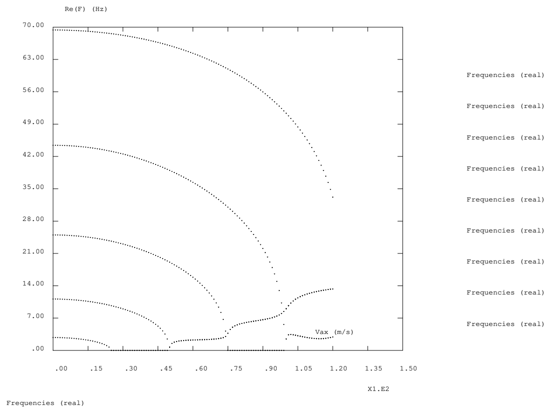
Velocity V = 0

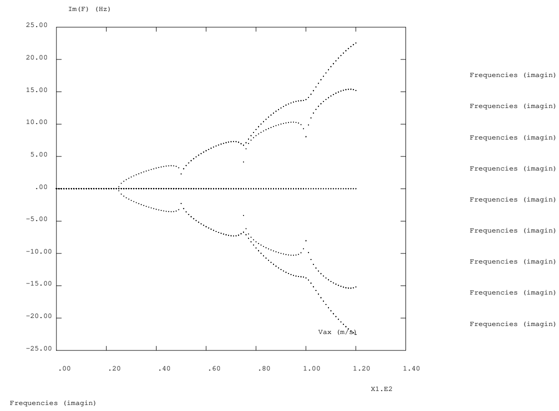
Mode	Frequency	Damping	real part	imag. part	stability
1	2.78	$-7.9.10^{-16}$	$1.4.10^{-14}$	17.5	STAB
2	11.11	$-7.0.10^{-16}$	$4.9.10^{-14}$	69.8	STAB
3	24.98	$1.0.10^{-16}$	$-1.6.10^{-14}$	157.0	STAB
4	44.41	$3.6.10^{-15}$	$-1.0.10^{-12}$	279.0	STAB
5	69.37	$-7.7.10^{-15}$	$3.4.10^{-12}$	435.8	STAB

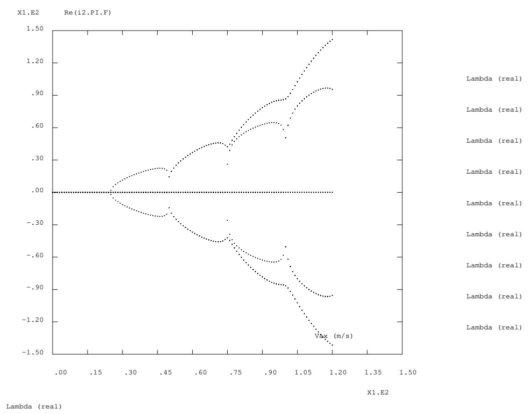
Velocity V = 120 m/s

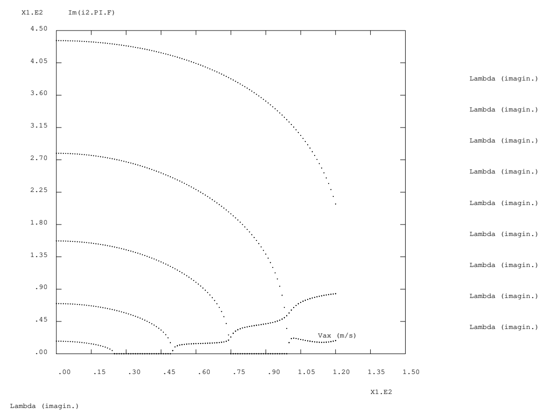
Mode	Frequency	Damping	real part	imag. part	stability
1	2.9	-5.2	95.4	18.3	INST
2	2.9	5.2	-95.4	18.3	STAB
3	13.3	1.7	-141.6	83.5	STAB
4	13.3	-1.7	141.6	83.5	INST
5	33.2	$1.2.10^{-12}$	$-2.5.10^{-10}$	208.4	STAB

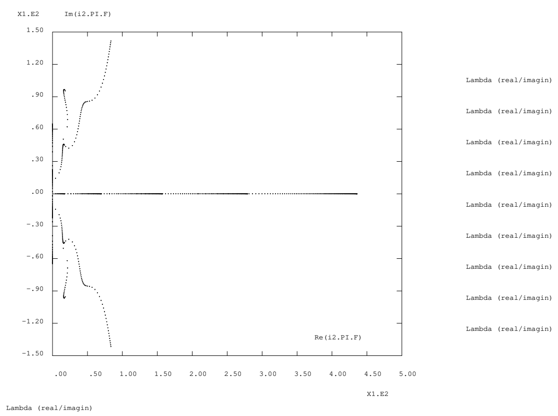
CASTEM figures











```
*          Test Vibr6.dgibi: Jeux de données
*          ---------------------------------
*
***************************************************
*     TEST : VIBR6                                *
*                                                 *
*   Modele de PAIDOUSSIS & ORTOJA-STARZEWSKI      *
*                                                 *
*       On recherche ici les modes propres        *
*   complexes d'un arbre soumis a un ecoulement   *
*   Arbre soumis a un ecoulement exterieur        *
*                axial permanent                  *
* Conditions aux limites : articule - articule    *
***************************************************
*
Opti Echo 0;
*
*--- Affichage
AFFICH = VRAI;
GRAPH = 'N';
IMPRPS = FAUX;
*
*Si (NON AFFICH);
*  Opti Echo 1;
*Finsi;
Si (IMPRPS);
  Opti Trac ps;
Finsi;
************************************
*   Caracteristiques du systeme    *
************************************
*--- Materiau
Rhostru = 4000.;
Youstru = 1.E10;
Nustru = 0.;
*--- Longueur du crayon
Lstru = 2.;
*--- Rayon du crayon
R = 10.E-3;
*--- Section du crayon
Sstru = PI*R*R;
*--- Masse lineique du crayon
Mstru = Rhostru*Sstru;
*--- Masse volumique du fluide
Rhof = 1000.;
*--- Masse ajoutee du fluide (fluide non confine)
Mfaj = Rhof*PI*R*R;
*--- Masse volumique equivalente
Rhoe = (Mstru+Mfaj)/Sstru;
*--- Vitesse axiale du fluide
Vmin = 0.;
Vmax = 120.;
PVa = 1.;
*
LVa = Prog Vmin PAS PVa Vmax;
*
**************************************
*         Maillage du systeme        *
**************************************
*
NbN = 100;
Mess 'Nombre de noeuds :' NbN;
Mess ' ';
Opti Dime 3 Elem Seg2;
PA = (-.5*Lstru) 0. 0.;
PB = (.5*Lstru) 0. 0.;
L1 = Droi NbN PA PB;
*
```

```
* Modl1 = Modl L1 Mecanique Elastique Isotrope Tuya;
* Matr1 = Matr Modl1 Youn Youstru Nu Nustru Rho Rhoe
*         Epai R Rayo R;
  *
  **************************************
  *   Masses, Rigidites ... du systeme *
  **************************************
  *
  *--- Masse
  Mstru = Masse Modl1 Matr1;
  *--- Raideur structrale
  Kstru = Rigi Modl1 Matr1;
  *--- Conditions aux limites : Probleme plan
  Cl1 = Bloq Ux L1;
  Cl2 = Bloq Uy L1;
  Cl3 = Bloq Rx L1;
  Cl4 = Bloq Rz L1;
  Cl5 = Bloq Uz (PA et PB);
  *
  Kstru = Kstru et Cl1 et Cl2 et Cl3 et Cl4 et CL5;
  *
  ***********************************
  *    Rigidites supplementaires    *
  ***********************************
  *
  Lpout = Lstru/NbN;
  *
  *--- Amortissement du a l'écoulement
  PAfu =
  (Prog -.5 0. 0. (Lpout/10.) .5 0. 0. (-1.*Lpout/10.) );
  PAfu = PAfu et
  (Prog 0. -.5 (-1.*Lpout/10.) 0.  0. .5 (Lpout/10.) 0.);
  PAfu = PAfu et
  (Prog 0. (Lpout/10.) 0. 0. 0. (-1.*Lpout/10.)
  (-1.*Lpout*Lpout/60.) 0.);
  PAfu = PAfu et
  (Prog (-1.*Lpout/10.) 0. 0. 0. (Lpout/10.) 0. 0.
  (-1.*Lpout*Lpout/60.));
  PAfu = PAfu et
  (Prog -.5 0. 0. (-1.*Lpout/10.) .5 0. 0. (Lpout/10.) );
  PAfu = PAfu et
  (Prog 0. -.5 (Lpout/10.) 0. 0. .5 (-1.*Lpout/10.) 0. );
  PAfu = PAfu et
  (Prog 0. (-1.*Lpout/10.) (Lpout*Lpout/60.) 0. 0.
  (Lpout/10.) 0. 0. );
  PAfu = PAfu et
  (Prog (Lpout/10.) 0. 0. (Lpout*Lpout/60.)
  (-1.*Lpout/10.) 0. 0. 0. );
  Afu = Manu Rigi type AMORTISSEMENT L1
        (MOTS UY UZ RY RZ) PAfu;
  *
  *--- Raideur due a l'ecoulement
  PKfu =
  (Prog (-6./(5.*Lpout)) 0. 0. -1.1 (6./(5.*Lpout)) 0. 0.
                                                       -.1);
  PKfu = PKfu et
  (Prog 0. (-6./(5.*Lpout)) 1.1 0. 0. (6./(5.*Lpout)) .1
                                                        0.);
  PKfu = PKfu et
  (Prog 0. .1 (-2.*Lpout/15.) 0. 0. -.1 (Lpout/30.) 0. );
  PKfu = PKfu et
  (Prog -.1 0. 0. (-2.*Lpout/15.) .1 0. 0. (Lpout/30.) );
  PKfu = PKfu et
  (Prog (6./(5.*Lpout)) 0. 0. .1 (-6./(5.*Lpout)) 0. 0.
                                                     1.1);
  PKfu = PKfu et
  (Prog 0. (6./(5.*Lpout)) -.1 0. 0. (-6./(5.*Lpout))
  -1.1                                                0. );
```

```
PKfu =
PKfu et (Prog 0. .1 (Lpout/30.) 0. 0. -.1
        (-2.*Lpout/15.) 0.   );
PKfu = PKfu et
(Prog -.1 0. 0. (Lpout/30.) .1 0. 0. (-2.*Lpout/15.) );
Kfu = Manu Rigi type RIGIDITE L1 (MOTS UY UZ RY RZ)
      PKfu;
*
*
*****************************************
*             Analyse modale            *
*****************************************
*
PMod = Prog 2.77 11.1 24.99 44.43 69.42;
Basr = VIBR PROCH PMod Kstru Mstru TBAS;
nnor (Basr.'MODES') (mots 'UZ');
*
Basm = Basr.'MODES';
Nbmod = DIME Basm-2;
Si (Affich);
Mess ' ';
Mess 'Fluide a l arret';
Mess '    Mode     Frequence ';
Mess '---------------------------';
  Repeter LMod Nbmod;
    i=&LMod;
    Mess '    ' i '    ' Basm.i.'FREQUENCE';
  Fin LMod;
  Mess ' ';
  Mess ' ';
  Mess ' ';
Finsi;
*
*****************************************
* Mise en forme du tableau de resultats *
*****************************************
*
Mod = Table;
Mod.Reel = Table;
Mod.Imag = Table;
Repeter Bloc6 (2*Nbmod);
  i=&Bloc6;
  Mod.Reel.i=Prog;
  Mod.Imag.i=Prog;
Fin Bloc6;
*
*****************************************
*        Projection des matrices        *
*****************************************
*
*--- Matrice masse
Mt = (PJBA MStru Basr);
*--- Matrices amortissement
Afp = (PJBA Afu Basr);
*--- Matrices de raideur
Ksp = (PJBA Kstru Basr);
Kfp = (PJBA Kfu Basr);
*
*--- Reperage de la vitesse critique
VSauv = 'N';
* ==================================== *
*      Boucle sur la vitesse axiale    *
* ==================================== *
*
Repeter Bloc10 (DIME LVa);
  k=&Bloc10;
  Va = Extr LVa k;
  Si (AFFICH);
  Mess 'Iteration' k;
  Mess 'Vitesse axiale : ' Va ;
  Mess ' ';
  Mess '---------------------------------------------------------------
--------------';
  Mess '  Mode     Frequence      Amort.      P. Reelle    P. Imaginaire
   Stabilité  ';
  Finsi;
*
*****************************************
*        Masses, rigidites ...          *
*****************************************
*
*---- Amortissement
  At =  (2*Mfaj*Va*Afp);
*---- Rigidite
  Kt = (Ksp) et ((Mfaj*Va*Va)*Kfp);
*
*****************************************
*      Calcul des modes complexes       *
*****************************************
*
  Basc = VIBC Mt Kt At;
  Basm = Basc.'MODES';
*
*--- Extraction des frequences
  Prf = Prog;
  Pif = Prog;
  Repeter Bloc2 (2*Nbmod);
    i=&Bloc2;
    Modn = Basm.i;
    f = Modn.'FREQUENCE_REELLE';
*    Si (>EG f 0.);
      Prf = Prf et (Prog Modn.'FREQUENCE_REELLE');
      Pif = Pif et (Prog Modn.'FREQUENCE_IMAGINAIRE');
*    Finsi;
  Fin bloc2;
*
*--- Tri croissant des frequences
  n1 = DIME Prf;
  Prfc = Prog (extr Prf 1);
  Pifc = Prog (extr Pif 1);
  Repeter Bloc3 (n1-1);
    i=&Bloc3+1;
    f = extr Prf i;
    n2 = DIME Prfc;
    Pos = 1;
    Repeter Bloc4 n2;
      j = &Bloc4;
      f1 = extr Prfc j;
      Si (> f f1);
        Pos = Pos + 1;
      Finsi;
    Fin Bloc4;
    Prfc = INSE Prfc Pos f;
    Pifc = INSE Pifc Pos (extr Pif i);
  Fin Bloc3;
*--- Sauvegarde
  Repeter Bloc5 n1;
    i=&Bloc5;
    f =  (Extr Prfc i);
    if = (Extr Pifc i);
    Si (AFFICH);
      Msg = 'STAB.';
      Si ((< if -1.D-10) et (>EG f 0.));
        Msg = 'INST.';
        Si (EGA VSauv 'N');
          Vcrit = Va - (PVa/2.);
```

```
          VSauv = 'O';
        Finsi;
      Finsi;
      Si (> f 0.);
*Valeurs propres relatives a lambda = i omega = ix2PIxf
        Mess  i  f  (if/f)  (-2*PI*if)  (2*PI*f) Msg;
      Sinon;
        Si (EGA f 0.);
          Mess i f
          '     -----     ' (-2*PI*if) (2*PI*f) Msg;
        Finsi;
      Finsi;
    Finsi;
    Mod.Reel.i = Mod.Reel.i et (Prog f);
    Mod.Imag.i = Mod.Imag.i et (Prog if);
  Fin Bloc5;
  Si (AFFICH);
  Mess '-------------------------------------------------------------------------
--------------';
  Mess ' ';
  Finsi;
*
Fin Bloc10;
*
**********************************
*           Traces               *
**********************************
*
*
Si ( EGA GRAPH 'O');
*
Nbcour=2*Nbmod;
Tab1 = Table;
Repeter Ident1 Nbcour;
  k=&Ident1;
  Tab1.k = 'NOLI';
Fin Ident1;
*
  k=1;
*--- Traces relatifs a la frequence
  Titr 'Frequences (partie reelle)';
  Evol1 = Evol BLAN manu Lva (Mod.Reel.k);
  Titr 'Frequences (partie imaginaire)';
  Evol2 = Evol BLAN manu Lva (Mod.Imag.k);
  Titr 'Lambda (partie reelle)';
*--- Traces relatifs a lambda = i 2 Pi f
  Evol3 = Evol BLAN manu Lva (-2.*PI*(Mod.Imag.k));
  Titr 'Lambda (partie imaginaire)';
  Evol4 = Evol BLAN manu Lva (2*PI*(Mod.Reel.k));
  Titr 'Lambda (partie reelle / partie imaginaire)';
  Evol5 = Evol BLAN manu (2.*PI*(Mod.Reel.k))
                             (-2.*PI*(Mod.Imag.k));
  Repeter Trac1 (Nbcour-1);
    k=&Trac1+1;
    Si (> (DIME Mod.Reel.k) 0);
    Titr 'Frequences (partie reelle)';
    Evol1 = Evol1 et (Evol BLAN manu Lva (Mod.Reel.k));
    Titr 'Frequences (partie imaginaire)';
    Evol2 = Evol2 et (Evol BLAN manu Lva (Mod.Imag.k));
    Titr 'Lambda (partie reelle)';
    Evol3 = Evol3 et (Evol BLAN manu Lva
                                (-2.*PI*(Mod.Imag.k)));
    Titr 'Lambda (partie imaginaire)';
    Evol4 = Evol4 et (Evol BLAN manu Lva
                                (2.*PI*(Mod.Reel.k)));
    Titr 'Lambda (partie reelle / partie imaginaire)';
    Evol5 = Evol5 et (Evol BLAN manu
           (2.*PI*(Mod.Reel.k)) (-2.*PI*(Mod.Imag.k)));
    Finsi;
  Fin Trac1;
  Dess Evol1 XBOR 0. 150. YBOR 0. 70. Lege
                    Titx 'Vax (m/s)'
                    Tity 'Re(F) (Hz)' Tab1;
  Dess Evol2 Lege
                    Titx 'Vax (m/s)'
                    Tity 'Im(F) (Hz)' Tab1;
  Dess Evol3 XBOR 0. 150. YBOR -150. 150. Lege
                    Titx 'Vax (m/s)'
                    Tity 'Re(i2.PI.F)' Tab1;
  Dess Evol4 XBOR 0. 150. YBOR 0. 450. Lege
                    Titx 'Vax (m/s)'
                    Tity 'Im(i2.PI.F)' Tab1;
  Dess Evol5 XBOR 0. 500. YBOR -150. 150. Lege
                    Titx 'Re(i2.PI.F)'
                    Tity 'Im(i2.PI.F)' Tab1;
Finsi;
*
**********************************
*   Test de bon fonctionnement   *
**********************************
*
Vth = (Pi/Lstru)*((Youstru*Pi*(R**4)/(4*mfaj))**0.5);
*Mess 'Erreur : ' (Abs((Vcrit-Vth)/Vth));
*Opti donn 5;
Si (Abs((Vcrit-25.)/25.) < 0.05);
  Erre 0;
Sinon;
  Erre 5;
Finsi;
*
FIN;
```

Test vibr6 **Comments**

1. **Creation of added stiffnesses**

   Afu = Manu Rigi type AMORTISSEMENT L1 (MOTS UY UZ RY RZ) PAfu;
   Kfu = Manu Rigi type RIGIDITE L1 (MOTS UY UZ RY RZ) PKfu;

   The operator MANU followed by RIGI keyword creates a RIGIDITE subtype object in which all elementary stiffness matrices are similar :
   - the elementary matrix is given by a list of reals (PAfu and PKfu),
   - the geometrical object is given by L1,
   - the type of stiffness is specified (AMORTISSEMENT and RIGIDITE),
   - the name of the components for each node of L1 is given by a list of words (MOTS UY UZ RY RZ).
   In this case, each elementary matrix is defined by a 8X8 matrix : 2 x (UY UZ RY RZ).

2. **Projection on modal basis**

   * Mass matrix
   Mt = (PJBA MStru Basr);
   * Damping matrices
   Afp = (PJBA Afu Basr);
   * Rigidity matrices
   Ksp = (PJBA Kstru Basr);
   Kfp = (PJBA Kfu Basr);

   The PJBA operator calculates the projection of stiffness objects (MStru, Afu, Kstru and Kfu) on an eigen mode basis (Basr).

3. **Computation of compound eigen shapes**

   Basc = VIBC Mt Kt At;
   Basm = Basc.'MODES';
   ...
   Modn = Basm.i;
   Prf = Prf et (Prog Modn.'FREQUENCE_REELLE');
   Pif = Pif et (Prog Modn.'FREQUENCE_IMAGINAIRE');

   VIBC computes the complex eigen values and eigen vectors of the general equation :
   $M\ddot{x} + C\dot{x} + Kx = 0$.
   The mass (Mt), stiffness (Kt) and damping (At) are given in matrices previously expressed relatively to a real eigen mode basis. If these matrices have not their subtypes corresponding with MASSE, RIGIDITE and AMORTISSEMENT, the input order is very important because they can't be sorted: they then are processed in order of their input.

   The generated object is a BASE_MODALE subtype table (Basc).
   Basc.'MODES' is a BASE_DE_MODES subtype table (Basm).
   Basm.i is a MODE_COMPLEXE subtype table (Modn) :

   Modn.'FREQUENCE_REELLE' : real part of the eigen frequency.
   Modn.'FREQUENCE_IMAGINAIRE' : imaginary part of the eigen frequency.
   Modn.'NUMERO_MODE' : number of the mode.
   Modn.'DEFORMEE_MODALE_REELLE' : real part of the eigen vector.
   Modn.'DEFORMEE_MODALE_IMAGINAIRE' : imaginary part of the eigen vector.

4. **Options for evolution display**

```
Dess Evol1 XBOR 0. 150. YBOR 0. 70. Lege
           Titx 'Vax (m/s)' Tity 'Re(F) (Hz)' Tab1;
```

The XBOR and YBOR keywords impose boundaries on X-axis (mini 0. and maxi 150.) and on Y-axis (mini 0. and maxi 70.).
The LEGE keyword imposes a caption for the different plotted curves. It imposes a square frame and the maximal number of curves is nine.
The TITX and TITY keywords allow to modify the names of abscissas and ordinates.

12.6 Test vibr10 Description sheet

Test name vibr10

Calculation type MECHANICS DYNAMICS
MASS UNBALANCE RESPONSE

Finite element type

Topic
Mass unbalance response.

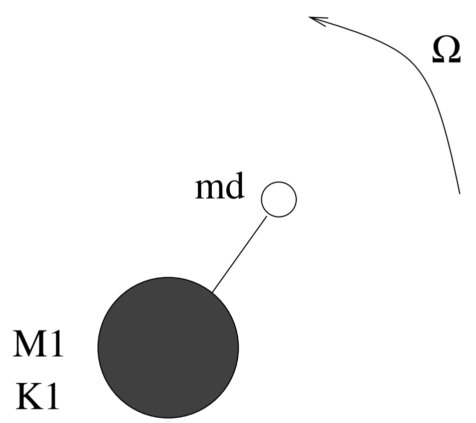
The system is represented by a point (mass M1, stiffness K1). A mass unbalance is added in the equation under the shape of a force.
The system to be solved is : $M\ddot{x} + C\dot{x} + Kx = F$ with F defined by $F = F_s.sin(\Omega.t) + F_c.cos(\Omega.t)$
The response is computed with VIBC operator and its option BALOUR.

Goal Find the small and big axes of the ellipse described by the system.

Reference Analytical solution.

Version 97' customer version

Model description



$$
\begin{aligned}
M1 &= 40kg \\
K1 &= 4.10^5 N/m \\
md &= 10^{-4} kg.m \\
\Omega &= 19.0985 rad/sec
\end{aligned}
$$

Results

X_{theor} (m)	$9.46402.10^{-8}$
X_s (m)	$9.46402.10^{-8}$
X_c (m)	$9.46402.10^{-8}$

```
*          Test Vibr10.dgibi: Jeux de données
*          ---------------------------------
*
**************************************************
*  TEST : VIBR10                                 *
*                                                *
*   Calcul de la reponse au balourd              *
*   Test de l'option 'BALOU de l'operateur       *
*   VIBC                                         *
**************************************************
OPTI 'DIME' 3 'ELEM' SEG2 ;
*
*donnees
P1 =  1. 0. 0. ;
m1 = 40. ;
k1 = 4.e5 ;
f1 = ((k1 / m1) ** 0.5) / (2. * pi) ;
MESS 'f1 =' f1 ;
*
TR1 = MANU 'POI1' P1 ;
*
*Matrices de masse et de rigidite :
MMass = MANU 'RIGI' TR1 (MOTS 'UY' 'UZ')
        (PROG m1 0. 0. m1) ;
MRigi = MANU 'RIGI' TR1 (MOTS 'UY' 'UZ')
        (PROG k1 0. 0. k1) ;
*
*deformees modales :
Phi1 = MANU 'CHPO' TR1 2 'UY' 1. 'UZ' 0. ;
Phi2 = MANU 'CHPO' TR1 2 'UY' 0. 'UZ' 1. ;
*
*points reperes des modes
PR1 = 0. 0. 0. ;
PR2 = 0. 0. 0. ;
*
*base des mods reels :
TBasR = TABLE 'BASE_MODALE' ;
TBasR.MODES = TABLE 'BASE_DE_MODES' ;
TBasR.MODES . 'MAILLAGE' = TR1 ;
TBasR.MODES . 1 = TABLE 'MODE' ;
TBasR.MODES . 1 . 'NUMERO_MODE' = 1 ;
TBasR.MODES . 1 . 'POINT_REPERE' = PR1 ;
TBasR.MODES . 1 . 'FREQUENCE' = f1 ;
TBasR.MODES . 1 . 'MASSE_GENERALISEE' = m1 ;
TBasR.MODES . 1 . 'DEPLACEMENTS_GENERALISES' =
  k1 - (2 * pi * f1) ;
TBasR.MODES . 1 . 'DEFORMEE_MODALE' = Phi1 ;
TBasR.MODES . 2 = TABLE 'MODE' ;
TBasR.MODES . 2 . 'NUMERO_MODE' = 2 ;
TBasR.MODES . 2 . 'POINT_REPERE' = PR2 ;
TBasR.MODES . 2 . 'FREQUENCE' = f1 ;
TBasR.MODES . 2 . 'MASSE_GENERALISEE' = m1 ;
TBasR.MODES . 2 . 'DEPLACEMENTS_GENERALISES' =
  k1 - (2 * pi * f1) ;
TBasR.MODES . 2 . 'DEFORMEE_MODALE' = Phi2 ;
*
*vitesse (rad/s)
omeg1 = 19.0985 ;
*
*-projection des matrices sur la base réelle :
  MMasP = PJBA TBasR MMass ;
  MRigP = PJBA TBasR MRigi ;
*
*-calcul de la force d'excitation du balourd
  MxD = 1e-4 ;
  Fy1s = MxD * (omeg1 ** 2) ;
  Fz1s = (-1) * MxD * (omeg1 ** 2) ;
  Fy1c = MxD * (omeg1 ** 2) ;
*   Fz1c = MxD * (omeg1 ** 2) ;
* *
*   F1s = MANU 'CHPO' P1 2 'FY' Fy1s 'FZ' Fz1s ;
    F1c = MANU 'CHPO' P1 2 'FY' Fy1c 'FZ' Fz1c ;
    PjF1s = PJBA F1s TBasR ;
    PjF1c = PJBA F1c TBasR ;
  *
    GEOx1 = EXTR PjF1s 'MAIL' ;
    GEOx2 = GEOx1 ;
  *
    XVal1 = PROG ;
    XVal2 = PROG ;
    REPE bbb2 (NBEL GEOx1) ;
     i2 = &bbb2 ;
     XP1 = 0. 0. 0. ;
     GEOx2 = GEOx2 ET XP1 ;
     Px1 = POIN GEOx1 i2 ;
     XVal1 = XVal1 ET (PROG (EXTR PjF1s 'FALF' Px1)) ;
     XVal2 = XVal2 ET (PROG (EXTR PjF1c 'FALF' Px1)) ;
    FIN bbb2 ;
    XVal2 = XVal1 ET XVal2 ;
  *
    PjFBal1 = MANU CHPO GEOx2 1 'FALF' XVal2 ;
  *
  *-calcul de la reponse :
    RepRe1 RepIm1 = VIBC 'BALOU' MMasP MRigP TBasR
                    PjFBal1 omeg1 ;
  *
  *-calcul des pt et gd axes de l'ellipse :
     YR1 = EXTR RepRe1 'UY' P1 ;
     ZR1 = EXTR RepRe1 'UZ' P1 ;
     MESS YR1 ZR1 ;
  *
  *-resultats attendus :
  An1= (MxD * (omeg1 ** 2)) / (k1 - ((omeg1 ** 2) * m1));
  MESS 'resultats attendus : ' An1 ;
  *
  FIN ;
```

Test vibr10 **Comments**

1. **Mass unbalance excitation**

```
F1s = MANU 'CHPO' P1 2 'FY' Fy1s 'FZ' Fz1s ;
F1c = MANU 'CHPO' P1 2 'FY' Fy1c 'FZ' Fz1c ;
PjF1s = PJBA F1s TBasR ;
PjF1c = PJBA F1c TBasR ;
GEOx1 = EXTR PjF1s 'MAIL' ;
GEOx2 = GEOx1 ;
*
XVal1 = PROG ;
XVal2 = PROG ;
REPE bbb2 (NBEL GEOx1) ;
i2 = &bbb2 ;
XP1 = 0. 0. 0. ; GEOx2 = GEOx2 ET XP1 ;
Px1 = POIN GEOx1 i2 ;
XVal1 = XVal1 ET (PROG (EXTR PjF1s 'FALF' Px1)) ;
XVal2 = XVal2 ET (PROG (EXTR PjF1c 'FALF' Px1)) ;
FIN bbb2 ;
XVal2 = XVal1 ET XVal2 ;
*
PjFBal1 = MANU CHPO GEOx2 1 'FALF' XVal2 ;
```

The mass unbalance force (PjFBal1) is expressed relatively to a real eigen mode basis. It is a CHAMPOIN object with one dual component FALF.
It must be obtained by the following way :
- Both components F_s and F_c are separated into two CHAMPOIN objects (here F1s and F1c).
- Those two CHAMPOIN objects are projected on the n real eigen mode basis (here TBasR basis is composed of 2 eigen modes). The projected objects are PjF1s and PjF1c.
- Then PjF1s and PjF1c are combined in order to obtain a 2 x n vector that is to say a CHAMPOIN object composed by one component FALF and supported by 4 points (GEOx2)

2. **Computation of the response**

```
RepRe1 RepIm1 = VIBC 'BALOU' MMasP MRigP TBasR PjFBal1 omeg1 ;
```

The operator VIBC followed by BALOU keyword computes the complex solutions in the case of an unbalanced excitation. It creates two CHAMPOIN objects (RepRe1 and RepIm1) which contain the displacement vectors :
- RepRe1 deals with $Cos(\Omega.t)$ components, and the real components.
- RepIm1 deals with $Sin(\Omega.t)$ components, and the imaginary components.

12.7 Test dyna12 Description sheet

Test name dyna12

Calculation type MECHANICS DYNAMICS 3D
TRANSIENT RESPONSE

Finite element type BARR

Topic Transient response of a launcher.

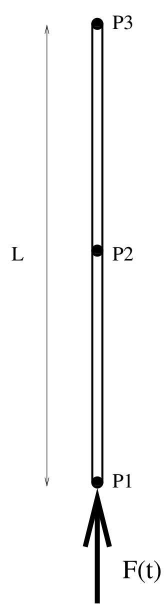
The launcher is submitted to an axial load during a limited period. Considering rigid body movements and vibration of the structure, the displacement history is searched for the bottom point.

Goal Find the displacement history of the bottom point (P1).

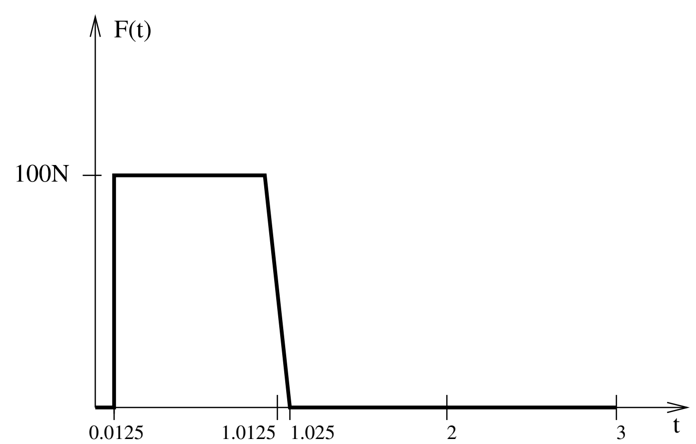
Reference J. S. Przemieniecki : Theory of matrix structural analyses (New York : McGraw-Hill Book Co., 1968) p340.

Version 97' customer version

Model description



$L = 140$
$A = 1$
$E = 1.10^4$
$\nu = 0.3$
$\rho = 0.1$



Results

	Analytic	CASTEM	Margin
UY(t=1.9125) Point P1	10.9	10.660	-2.20%
UY(t=2.0625) Point P1	11.732	11.289	-3.78%

```
*          Test Dyna12.dgibi: Jeux de données       *
*          ---------------------------------       *
*                                                  *
GRAPH='N';
SAUT PAGE ;
*******************************************************
*   REPONSE TRANSITOIRE D 'UNE FUSEE -METHODE DIRECT
*******************************************************
*
* DESCRIPTION DU PROBLEME
* -----------------------
*
* UNE FUSEE EST SOUMISE A UN CHARGEMENT AXIAL POUR UNE
* DUREE LIMITEE. IL S'AGIT DE TROUVER LES HISTORIQUES
* DE DEPLACEMENTS AUX EXTREMITES ET AU MILIEU DE LA
* FUSEE CONSIDERANT LE MOUVEMENT DE CORPS RIGIDE ET
* LA VIBRATION DE LA STRUCTURE .
*
* REFERENCE :
* ----------
*
* J. S. PRZEMIENIECKI,THEORY OF MATRIX STRUCTURAL
* ANALYSES (NEW YORK: MCGRAW-HILL BOOK CO.,1968)P.340
*
*******************************************************
OPTION  ECHO 0 ;
OPTIO DIME 3 ELEM SEG2 ;
*
*  MAILLAGE
*
 P1 = 0. 0. 0. ;P2 = 0. 70. 0. ;P3 = 0. 140. 0. ;
 L1 = D 1 P1 P2 ; L2 = D 1 P2 P3 ;LTOT = L1 ET L2 ;
*
*  MODELE
*
 MOD1 = MODELE LTOT MECANIQUE ELASTIQUE BARR ;
*
*  CARACTERISTIQUES MATERIELLES ET GEOMETRIQUES
*
 MAT1 = MATER MOD1 YOUN 1.E4  NU 0.3 RHO 0.1 ;
 CAR1 = CARAC MOD1 SECT 1. ;
*
*  MATRICE DE RIGIDITE
*
 RIG1 = RIGIDITE MOD1 (MAT1 ET CAR1) ;
*
*  MATRICE DE MASSE
*
 MAS1 = MASSE MOD1 (MAT1 ET CAR1)  ;
*
* CHARGEMENT
*
LI1 = PROG 0. 0.0125 1.0125 1.025  3. ;
LI2 = PROG 0. 1. 1. 0.  0.   ;
F1 = FORCE (0. 100. 0.) P1   ;
EV = EVOL MANU T LI1 F(T) LI2 ;
CHA1 = CHAR 'FORC' F1  EV ;
*
*  PREPARATION DE LA TABLE DE DONNEES POUR DYNAMIC
*
LIS1 = PROG 1.9125 2.0625  ;
TAB1 = TABLE ;
TAB1.'DEPL' = MANU CHPO LTOT 3 UZ 0. UY 0. UX 0.  ;
TAB1.'VITE' = MANU CHPO LTOT 3 UZ 0. UY 0. UX 0. ;
TAB1.'CHAR' = CHA1 ;
TAB1.'RIGI' = RIG1 ;
TAB1.'MASS' = MAS1 ;
TAB1.'FREQ' = 20. ;
TAB1.'INST' = LIS1 ;
TAB2 = DYNAMIC TAB1 ;
*
* COMPARAISON AVEC LES RESULTATS ANALYTIQUES
*
 DEP1 = TAB2 . 1 . DEPL ;
 U1 = EXTR DEP1 P1 UY ; U1TH = 10.89972 ;
 ERR1 = ((U1 - U1TH) / (ABS U1TH)) * 100. ;
 DEP2 = TAB2 . 2 . DEPL ;
 U2 = EXTR DEP2 P1 UY ; U2TH = 11.73228  ;
 ERR2 = ((U2 - U2TH) / (ABS U2TH)) * 100. ;
MESS ' DEPLACEMENT VERTICAL DE LA BASE    T= 1.9125'  ;
'MESS''THEORIE  'U1TH'CASTEM  'U1 '%ERREUR  'ERR1  ;
MESS ' DEPLACEMENT VERTICAL DE LA BASE    T= 2.0625'  ;
'MESS''THEORIE  'U2TH'CASTEM  'U2 '%ERREUR  'ERR2  ;
*
SI ((MAXI ABS (PROG ERR1 ERR2 )) < 5);
ERRE 0;
SINON;
ERRE 5;
FINSI;
FIN ;
```

Test dyna12 **Comments**

1. **Load as a function of time**

```
LI1 = PROG 0. 0.0125 1.0125 1.025 3. ;
LI2 = PROG 0. 1. 1. 0. 0. ;
F1 = FORCE (0. 100. 0.) P1 ;
EV = EVOL MANU T LI1 F(T) LI2 ;
CHA1 = CHAR 'FORC' F1 EV ;
```

The operator CHAR builds an object of type CHARGEMENT, which contains spatial and temporal descriptions of the load :
- the spatial description is given by the object F1,
- the temporal description is given by the object EV which is called evolution (built by the operator EVOL, EV describes the load (object LI2) as a function of the time (object LI1)).

2. **Use of DYNAMIC subroutine**

```
* INPUT DATA FOR DYNAMIC SUBROUTINE
*
LIS1 = PROG 1.9125 2.0625 ;
TAB1 = TABLE ;
TAB1.'DEPL' = MANU CHPO LTOT 3 UZ 0. UY 0. UX 0. ;
TAB1.'VITE' = MANU CHPO LTOT 3 UZ 0. UY 0. UX 0. ;
TAB1.'CHAR' = CHA1 ;
TAB1.'RIGI' = RIG1 ;
TAB1.'MASS' = MAS1 ;
TAB1.'FREQ' = 20. ;
TAB1.'INST' = LIS1 ;
TAB2 = DYNAMIC TAB1 ;
```

The operator DYNAMIC carries out step by step dynamic computations, using the Newmark centered algorithm. All required data are given in the object TAB1 defined by the following indexes :
-'DEPL' : initial displacements,
-'VITE' : initial velocity which components are the same as components of initial displacements,
-'CHAR' : the load (as a function of time),
-'RIGI' : the rigidity matrix,
-'MASS' : the mass matrix,
-'FREQ' : the cut off frequency which determines the time step,
-'INST' : the list of moments to save.
The value of time step will be equal to 0.25 divided by the cut off frequency (in this case 20 Hz) which gives a step of 0.0125 sec.

3. **Description of DYNAMIC result**

```
DEP1 = TAB2 . 1 . DEPL ; U1 = EXTR DEP1 P1 UY ;
DEP2 = TAB2 . 2 . DEPL ; U2 = EXTR DEP2 P1 UY ;
```

The object created by DYNAMIC subroutine is a table which contains as many tables as saved moments. Here TAB2 is composed of two tables because the list TAB1.INST contains two moments : TAB2 . 1 and TAB2 . 2.
Each table has three indexes :
- 'TEMP' : the value of the moment
- 'DEPL' : the displacement
- 'VITE' : the velocity
TAB2 . 1 . DEPL is a displacement field. A value can be extracted by using EXTR operator and specifying the desired point and the required component.

12.8 Test dyna11 Description sheet

Test name dyna11

Calculation type MECHANICS DYNAMICS 2D

Finite element type QUA4

Topic
Transient response of an internal pressure.

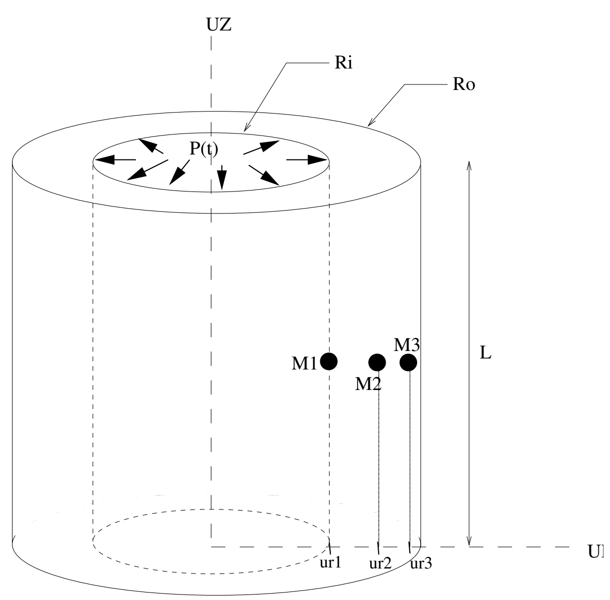
A thick-walled cylinder is suddenly subjected to an internal pressure. The problem consists in finding displacement maxima of some points.

Goal Find displacement maxima of three points located on the cylinder half-length.

Reference S. H.Crandall, N. C. Dahl and T. J. Lardner : An introduction to the mechanics of solids, 2nd ed. (New York : McGraw-Hill Book Company, 1972) pp. 293-297.

Version 97' customer version

Model description



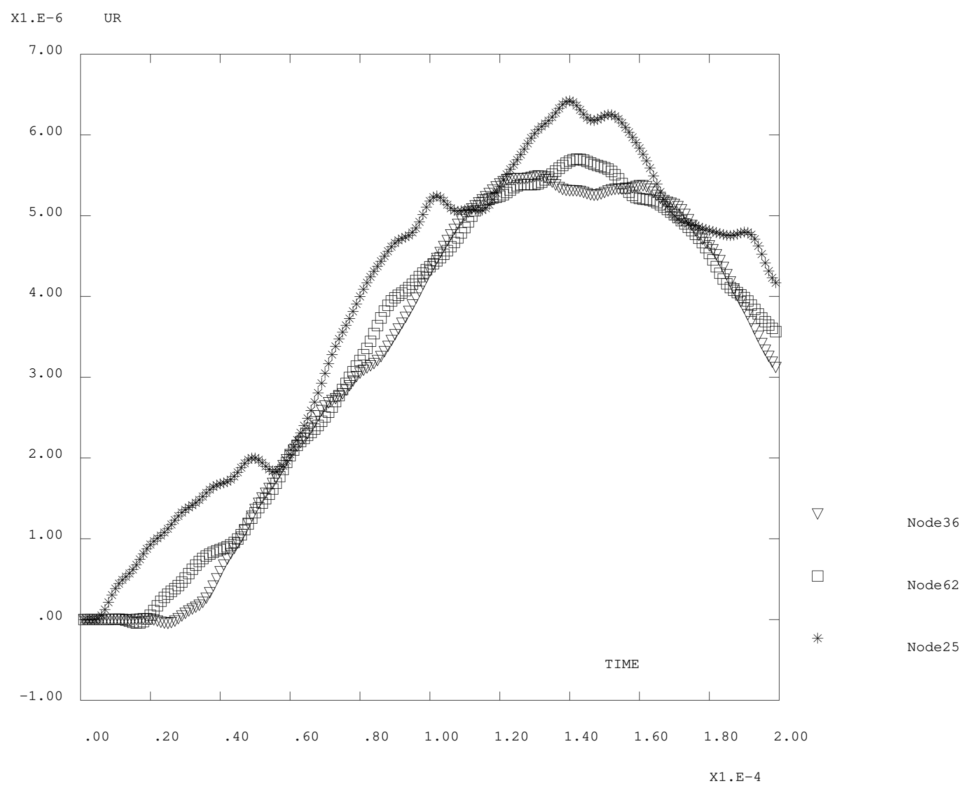
$$
\begin{aligned}
L &= 8in \\
R_i &= 6in \\
R_o &= 12in \\
ur_1 &= R_i \\
ur_2 &= 9in \\
ur_3 &= 11in \\
\rho &= 0.28lb/in^3 \\
1/g &= 1/(386.4in/sec^2) \\
&= 0.002588 \\
E &= 30.10^6 lb/in^2 \\
\nu &= 0.0 \\
P &= 10lb/in^2
\end{aligned}
$$

Test dyna11 **Results**

Results : Ratio transient/static

	Analytical	CASTEM	%Margin
M1	2	1.93	-3.46
M2	2	2.06	2.79
M3	2	2.05	2.74

CASTEM figures



RADIAL DISPLACEMENT THICK-WALLED CYLINDER

```
*          Test Dyna11.dgibi: Jeux de données
*          ---------------------------------
*
GRAPH='N';
SAUT PAGE ;
*******************************************************
* REPONSE TRANSITOIRE A UNE PRESSION INTEREN
*******************************************************
*
* DESCRIPTION DU PROBLEME
* -----------------------
* UN CYLINDRE A PAROIS EPAISSES EST SOUMISE BRUTALEMENT
* A UNE PRESSION INTERNE . IL S 'AGIT DE TROUVER LES
* PICS DES DEPLACEMENTS DANS CERTAINS POINTS DE LA
* STRUCTURE EN QUESTION .
*
* REFERENCE :
* ----------
* S. H. CRANDALL,N.C. DAHL,AND T. J. LARDNER ,AN
* INTRODUCTION TO THE MECHANICS OF SOLIDS,2ND ED.
* (NEW YORK :MCGRAW-HILL BOOK COMPANY,1972) PP. 293-297
*******************************************************
OPTION  ECHO 0 ;
OPTIO DIME 2 MODE AXIS ELEM QUA4 ;
*
*  MAILLAGE
*
 P1 = 6. 0. ; P2 = 12. 0. ; P3 = 12. 8. ; P4 = 6. 8. ;
 L1 = D 6 P1 P2 ; L2 = D 8 P2 P3 ;L3 = D 6 P3 P4 ;
 L4 = D 8 P4 P1 ; LTOT = L1 ET L2 ET L3 ET L4 ;
 S1 = SURF LTOT PLAN  ;
*
*  MODELE
*
 MOD1 = MODELE S1 MECANIQUE ELASTIQUE  ;
*
*  PROPRIETES DE MATERIAUX
*
 RO = 0.28 * 0.002588 ;
 MAT1 = MATER MOD1 YOUN 30.E6 NU 0.0 RHO RO ;
*
*  CONDITIONS AUX LIMITES
*
 CL1 = BLOQ L1 UZ ;CL2 = BLOQ L3 UZ ;
*
*  MATRICE DE RIGIDITE
*
 RIG1 = RIGIFITE MOD1 MAT1 ;
*
*  MATRICE DE MASSE
*
 MAS1 = MASSE MOD1 MAT1 ;
*
* CHARGEMENT
*
LI1 = (PROG 0. 1.E-6 2.E-6 3.E-6 4.E-6 ) ET
      (PROG  5e-6 pas 1e-6 204E-6 );
LI2 = (PROG 0. 0. 0. 0. 0. ) ET (PROG 200 * 1.) ;
F1 = PRES MOD1 'MASS' 10. L4  ;
EV = EVOL MANU T LI1 F(T) LI2 ;
CHA1 = CHAR 'PRES' F1  EV ;
*
*  PREPARATION DE LA TABLE DE DONNEES POUR DYNAMIC
*
LIS1 = PROG  0. PAS 1.E-6  200.E-6  ;
TAB2 = TABLE ;
TAB2.'DEPL' = MANU CHPO S1 2 UZ 0. UR 0.  ;
TAB2.'VITE' = MANU CHPO S1 2 UZ 0. UR 0. ;
* TAB2.'CHAR' = CHA1 ;
* TAB2.'RIGI' = RIG1 ;
* TAB2.'MASS' = MAS1 ;
TAB2.'FREQ' = 250000. ;
TAB2.'INST' = LIS1 ;
TAB1 = DYNAMIC TAB2 ;
*
*  EXTRAIRE  LES OBJETS D'EVOLUTION DE DEPLACEMENTS
*
LT = PROG ;  LUR1 = PROG  ;
LUR2 = PROG  ;
LUR3 = PROG  ;
I = 0 ;
M1 = NOEUD 25 ;M2 = NOEUD 62 ; M3 = NOEUD 36 ;
NBB = (DIME LIS1) - 1  ;
 REPETER  BB1 NBB  ;
  I = I + 1 ;
   LT = LT ET( PROG(TAB1 . I . 'TEMP' ));
   DEP1 = TAB1 . I . 'DEPL' ;
   LUR1 = LUR1 ET( PROG (EXTR DEP1 UR  M1) );
   LUR2 = LUR2 ET( PROG (EXTR DEP1 UR  M2) );
   LUR3 = LUR3 ET( PROG (EXTR DEP1 UR  M3) );
 FIN BB1 ;
EV1 =  EVOL MANU  'TIME' LT  'UR ' LUR1 ;
EV2 =  EVOL MANU  'TIME' LT  'UR ' LUR2 ;
EV3 =  EVOL MANU  'TIME' LT  'UR ' LUR3 ;
*
* ---  TRACER LES COURBES
*
 TABDD = TABLE ;
 TABDD.1 = 'MARQ ETOI NOLI '  ;
 TABDD.2 = 'MARQ CARR NOLI ' ;
 TABDD.3 = 'MARQ TRIA NOLI ' ;
 TABDD.'TITRE' = TABLE ;
 TABDD.'TITRE'. 1 = MOT 'N 25'  ;
 TABDD.'TITRE'. 2 = MOT 'N 62' ;
 TABDD.'TITRE'. 3 = MOT 'N 36' ;
 'TITR' 'RADIAL DISPLACEMENT  THICK-WALLED CYLINDER ' ;
  SI (NEG GRAPH 'N');
   DESS (EV1 ET EV2 ET EV3) TABDD 'LEGE' ;
 FINSI  ;
*
*   COMPARAISON AVEC LA THEORIE
*
 UR1 =  (MAXI LUR1) / 3.326E-6  ;  UR1TH = 2. ;
 ERR1 = ((UR1 - UR1TH) / (ABS UR1TH)) * 100. ;
 UR2 =  (MAXI LUR2) / 2.773E-6  ;  UR2TH = 2. ;
 ERR2 = ((UR2 - UR2TH) / (ABS UR2TH)) * 100. ;
 UR3 =  (MAXI LUR3) / 2.673E-6  ;  UR3TH = 2. ;
 ERR3 = ((UR3 - UR3TH) / (ABS UR3TH)) * 100. ;
MESS
'RAPPORT  TRANSITOIRE / STATIQUE       '   ;
MESS
'NODE 25   THEORIE 'UR1TH 'CASTEM  ' UR1 '%ERREUR 'ERR1;
MESS
'NODE 62   THEORIE 'UR2TH 'CASTEM  ' UR2 '%ERREUR 'ERR2;
MESS
'NODE 36   THEORIE 'UR3TH 'CASTEM  ' UR3 '%ERREUR 'ERR2;
*
SI ((MAXI ABS (PROG ERR1 ERR2 ERR3 )) < 5);
ERRE 0;
SINON;
ERRE 5;
FINSI;
FIN ;
```

Test dyna11 **Comments**

1. **Unit system**

```
RO = 0.28 * 0.002588 ;
MAT1 = MATER MOD1 YOUN 30.E6 NU 0.0 RHO RO ;
```

There is no unit system on CASTEM. Therefore data must be given in a right system according to the basic law of mechanics : $F = M.\gamma$.
It is necessary here to give the density in mass per volume unit that is to say to divide the density (given in lb/in^3) by the acceleration constant g (given in in/sec^2).

2. **Extracting of DYNAMIC results**

```
LT = PROG ; LUR1 = PROG ;
...
I = 0 ;
M1 = NOEUD 25 ;
...
NBB = (DIME LIS1) - 1 ;
REPETER BB1 NBB ;
I = I + 1 ;
LT = LT ET( PROG(TAB1 . I . 'TEMP' ));
DEP1 = TAB1 . I . 'DEPL' ;
LUR1 = LUR1 ET( PROG (EXTR DEP1 UR M1) );
...
FIN BB1 ;
EV1 = EVOL MANU 'TIME' LT 'UR ' LUR1 ;
```

To get the displacement evolution as a function of time, it is necessary to extract each saved instant and store it in a list of reals (LT). In the same way, each saved UR-displacement of the required point is stored in the LUR1 list of reals. Then an object of EVOLUTION type (EV1) can be obtained which represents the UR-displacement of M1 point as a function of time.

3. **Display of graph with a caption**

```
TABDD = TABLE ;
TABDD.1 = 'MARQ ETOI NOLI TITR Node25' ;
TABDD.2 = 'MARQ CARR NOLI TITR Node62' ;
TABDD.3 = 'MARQ TRIA NOLI TITR Node36' ;
TITR 'RADIAL DISPLACEMENT THICK-WALLED CYLINDER ' ;
DESS (EV1 ET EV2 ET EV3) TABDD 'LEGE' ;
```

To get a caption the LEGE keyword is added to the DESS operator, but it is necessary to indicate some specific options. These options are defined in the TABDD table. Each index of the table correspond to one curve : TABDD.1 is related to the first curve indicated in the DESS call.
The MARQ keyword followed by ETOI indicates that a symbol (a star) will be plotted as a marker at different points of the curve (CARR correspond to squares, TRIA correspond to triangles).
NOLI indicates that the segments connecting the points won't be plotted.
The TITR keyword followed by a string (maximum 20 letters and no gap between letters) defines the caption related to the required curve.

12.9 Test dy_devo5 Description sheet

Test name dy_devo5

Calculation type MECHANICS DYNAMICS 3D
TRANSIENT RESPONSE

Finite element type POUT

Topic
Transient response of a supported beam.

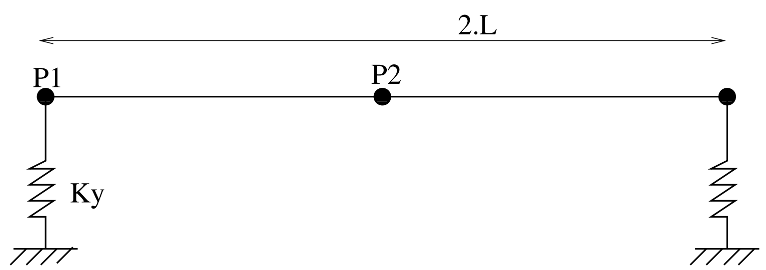
The beam is simply supported by elastic supports.The triangular distributed load has its amplitude as a function of time. The problem consists in finding the maximum deflection for different moments.

Goal Find the maximum deflection of the beam.

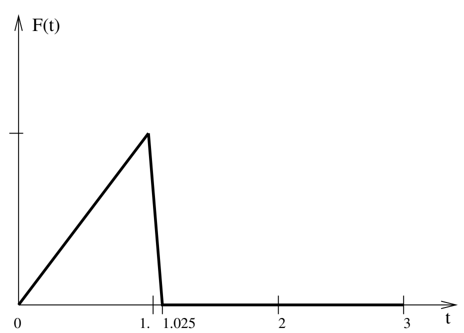
Reference Walter C. Hury and Moshe F. Rubinstein : Dynamics of structures (Englewood Cliffs : Prentice-Hall, Inc., 1964) p.294.

Version 97' customer version

Model description



$$
\begin{aligned}
L &= 50m \\
K_y &= (974.1/2)N/m \\
A &= 1m^2 \\
I_y &= 10m^4 \\
I_z &= 10m^4 \\
G &= 1.m^4 \\
E &= 10^6 Pa \\
\nu &= 0.3 \\
\rho &= 0.1kg/m^3 \\
F(x,t) &= \int_0^{2L} -(\tfrac{2x}{L}.F(t).S.\rho)dx
\end{aligned}
$$



Test dy_devo5 **Results**

Results

	Theoretical	CASTEM	%margin
ω (shape 1)	7.155 rad/s	7.1498 rad/s	0.007
ω (shape 2)	28.23 rad/s	29.679 rad/s	5.13
$deflection_{max}(t = 1.45s)$	0.23382 m	0.23361 m	0.009
$deflection_{max}(t = 1.9s)$	-0.24283 m	-0.2385 m	1.78

CASTEM figures

```
*        Test Dy_devo5.dgibi: Jeux de données
*        ------------------------------------
*
GRAPH='N';
SAUT PAGE ;
*******************************************************
*  REPONSE TRANSITOIRE D'UNE POUTRE -METHODE MODALE
*******************************************************
*
* DESCRIPTION DU PROBLEME
* -----------------------
*
* UNE POUTRE EST MONTEE SUR DES APPUIS ELASTIQUES .
* SUPPOSANT UNE CHARGE REPARTIE TRIANGULAIRE AVCE UNE
* INTENSITE VARIABLE DANS LE TEMPS, ON CHERCHE A
* CALCULER LA FLECHE MAXIMALE EN DIFFERENTS INSTANTS.
*
* REFERENCE :
* ----------
*
* WALTER C. HURY AND MOSHE F. RUBINSTEIN,DYNAMICS OF
* STRUCTURES, (ENGLEWOOD CLIFFS: PRENTICE-HALL,
* INC.,1964),P.294
*******************************************************
OPTION  ECHO 0 ;
OPTION DIME 3 ELEM SEG2 ;
*
*  MAILLAGE
*
 P1 = 0. 0. 0. ; P2 = 50. 0. 0. ;
 L1 = D 4 P1 P2  ;
*
* MODELE
*
 MOD1 = MODELE L1 MECANIQUE ELASTIQUE POUT ;
*
*  CARACTERISTIQUES MATERIELLES ET GEOMETRIQUES
*
 MAT1 = MATER MOD1 YOUN 1.E6 NU 0.3  RHO 0.1 ;
 CAR1 = CARAC MOD1 SECT 1 INRY 10. INRZ 10. TORS 1.
  VECT (0. 1. 0.)  ;
*
*   CONDITIONS AUX LIMITES : APPUI ELASTIQUE
*          ET CONDITIONS DE SYMETRIE
*
  K = 974.1 / 2  ;
 CL1 = APPUI UY K  P1  ;
 CL2 = BLOQ L1 RX RY UX UZ ;
 CL3 = BLOQ P2 RZ    ;
 CLTOT = CL1  ET CL2 ET CL3 ;
*
*  MATRICE DE  RIGIDITE
*
 RIG1 = RIGIDITE MOD1 (MAT1 ET CAR1 ) ;
*
*  MATRICE DE MASSE
*
 MAS1 = MASSE MOD1 (MAT1 ET CAR1 ) ;
*
*  CALCUL DES MODES
*
 TAB1 = VIBR  'INTERVALE' 0. 5.
   (RIG1 ET CLTOT) MAS1   'TBAS'  ;
TAB2 = TAB1.MODES ;
NNOR TAB2 AVEC (MOTS UY ) ;
NMOD = DIME TAB2  - 2 ;
IB = 0 ;
 REPETE BOUC NMOD ;
*   IB = IB + 1   ;
*   F1 = (TAB2.IB).'FREQUENCE'  ;
*   OMEG1 = (F1 * 2.) * PI ;
   SI (IB 'EGA' 1 ) ; OMEG1TH = 7.155 ;
   SINON ; OMEG1TH = 28.23   ; FINSI ;
 ERR1 = ((OMEG1 - OMEG1TH) / (ABS OMEG1TH)) * 100. ;
MESS
'MODE 'IB 'THEORIE 'OMEG1TH'CASTEM 'OMEG1 '%ERREUR '
 ERR1 ;
  DEP1 = (TAB2.IB).DEFORMEE_MODALE ;
  DEF1 = DEFORM DEP1 L1 1. ROUG ;
  DEF0 = DEFORM DEP1 L1 0.  ;
  SI (NEG GRAPH 'N');
    TRAC (0 0 10000)(DEF0 ET DEF1)  ;
  FINSI;
 FIN BOUC  ;
*
TABM = TAB1  ;
*
* --  PREPARATION DES DONNEES POUR L'OPERATEUR DYNE
*
*
* CHARGEMENT
*
*
*  FORCE REPARTIE
*
 FC1 = COOR 1 L1 ;
 FC2 = (FC1 / -50.) * 2. ;
 FC3 = NOMC FC2 UY ;
 MODB = MODEL L1 MECANIQUE ELASTIQUE BARR ;
 MATB  = MATER MODB SECT 1. RHO 1. ;
 MASB  = MASSE MODB MATB ;
 FC4 = MASB * FC3  ;
 FC5 = EXCO FC4 FY FY ;
LI1 = PROG 0.  1.  1.025  3.0 ;
LI2 = PROG 0.  1.    0.  0.  ;
EV = EVOL MANU T LI1 F(T) LI2 ;
CHA1 = CHAR 'FORC' FC5  EV ;
TABCHAR = TABLE 'CHARGEMENT' ;
*
*   PROJECTION SUR LA BASE MODALE
*
CHA2 = PJBA CHA1 TAB1 ;
TABCHAR.'BASE_A' = CHA2  ;
*
*  TABLE DE SORTIE
*
 TABSOR = TABLE 'SORTIE' ;
 TABVAR = TABLE 'VARIABLE' ;
 TABSOR.'VARIABLE' = TABVAR ;
 TABVAR.'DEPLACEMENT' = VRAI ;
 TABVAR.'VITESSE' = FAUX   ;
 TABVAR.'ACCELERATION' = FAUX  ;
*
* APPEL A DYNE DE_VOGELAERE
*
TABRES =
DYNE DE_VOGELAERE TABM TABCHAR TABSOR 80 0.025 1 ;
*
*  RECOMPOSITION POUR LES TEMPS 1.45 ET 1.9
*
DEP1 = RECO TABRES TABM 1.45 'DEPL'  ;
DEP2 = RECO TABRES TABM 1.9 'DEPL'  ;
*
*  COMPARAISON AVEC DES RESULTATS ANALYTIQUES
*
 FLECH1 = EXTR DEP1 P2 UY  ;FLECH1TH = 0.23382   ;
```

```
 ERR1 = (( FLECH1 - FLECH1TH) / (ABS FLECH1TH)) * 100.;
MESS '    FELECHE MAXIMALE  T = 1.45       '  ;
MESS'THEORIE   'FLECH1TH'CASTEM  'FLECH1'%ERREUR 'ERR1;
*
 FLECH2 = EXTR DEP2 P2 UY  ;FLECH2TH = -0.24283   ;
 ERR2 = (( FLECH2 - FLECH2TH) / (ABS FLECH2TH)) * 100.;
MESS '    FELECHE MAXIMALE  T = 1.9      '  ;
MESS'THEORIE   'FLECH2TH'CASTEM  'FLECH2'%ERREUR 'ERR2;
*
SI ((MAXI ABS (PROG ERR1 ERR2)) < 5);
ERRE 0;
SINON;
ERRE 5;
FINSI;
FIN ;
```

Test dy_devo5 **Comments**

1. **Norming of mode shapes**

   NNOR TAB2 AVEC (MOTS UY) ;

   The NNOR instruction norms the object TAB2 so that the greatest value equal 1. The keyword AVEC limits the research only to UY component.

2. **Distributed load**

   FC1 = COOR 1 L1 ;
   FC2 = (FC1 / -50.) * 2. ;
   FC3 = NOMC FC2 UY ;
   MODB = MODEL L1 MECANIQUE ELASTIQUE BARR ;
   MATB = MATER MODB SECT 1. RHO 1. ;
   MASB = MASSE MODB MATB ;
   FC4 = MASB * FC3 ;

   The distributed load is created by the multiplication of the load FC3 (which is a CHPOINT) by the mass matrix MASB. This method allows to balance the load in the meaning of finite elements.

3. **Projection of the load onto the modal basis**

   TABCHAR = TABLE 'CHARGEMENT' ;
   CHA2 = PJBA CHA1 TAB1 ;
   TABCHAR.'BASE_A' = CHA2 ;

   The PJBA operator projects the load CHA1 onto the modal basis TAB1. The projected load CHA2 is put in the table TABCHAR of CHARGEMENT subtype and indexed by BASE_A. The A-basis is the modal basis in which the equations are uncoupled.

4. **Declaration of the output table**

   * OUTPUT TABLE
   *
   TABSOR = TABLE 'SORTIE' ;
   TABVAR = TABLE 'VARIABLE' ;
   TABSOR.'VARIABLE' = TABVAR ;
   TABVAR.'DEPLACEMENT' = VRAI ;
   TABVAR.'VITESSE' = FAUX ;
   TABVAR.'ACCELERATION' = FAUX ;

   It is necessary to define the output table and its subtype : the TABSOR table of SORTIE subtype defines the data to be saved during the use of DYNE operator.
   TABSOR.VARIABLE is a table too, its subtype is VARIABLE.
   In this example, only the displacements will be saved because the content of the boolean variable (TABVAR.'DEPLACEMENT') is true and the content of the other boolean variables (TABVAR.'VITESSE' and TABVAR.'ACCELERATION') is false.

5. **Calling of DYNE De_Vogelaere operator**

   TABRES = DYNE DE_VOGELAERE TABM TABCHAR TABSOR 80 0.025 1 ;

   - The TABM table contains the modal basis.
   - The TABCHAR table contains the evolution of applied free forces (which are projected onto the modal base).
   - The TABSOR table defines the results requested in the TABRES table.
   - The number of steps is 80, the value of time step is 0.025.
   - A result is saved every 1 computation step.
   - TABRES contains the results of the computation.

   The maximum value of time step is about one tenth of the smallest period of the used mode shapes.

6. **Content of the result table TABRES**

   The TABRES subtype is RESULTAT_DYNE.
   - TABRES.'TEMPS_DE_SORTIE' contains the list of output times.
   - TABRES. i is a table containing the results of the i^{th} output step. The indices of TABRES. i depend on the requested variables given previously in TABVAR. Here we will find only the DEPLACEMENT index.

   TABRES. i .DEPLACEMENT is an object of CHPOINT type which component is ALFA because the computation was done in A-basis.

7. **Recombination of TABRES table for moments 1.45 and 1.9**

   DEP1 = RECO TABRES TABM 1.45 'DEPL' ;
   DEP2 = RECO TABRES TABM 1.9 'DEPL' ;

   The RECO operator recombines the modes and the static solutions contained in a modal basis from the modal contributions. The requested time is indicated (here 1.45 then 1.9) with the name of the variable (here DEPLACEMENT). DEP1 and DEP2 are CHPOINT objects which can be used like results of a static computation.

12.10 Test dy_devo6 Description sheet

Test name dy_devo6

Calculation type MECHANICS DYNAMICS 3D
IMPACT, LINKAGE POINT-PLANE

Topic
Impact of a great mass against a plane.

A great mass with a sinusoidal movement (composed by two different amplitudes) impacts a plane.

The computation is done with a linkage point-plane and with taking into consideration the plastification of the impact spring.

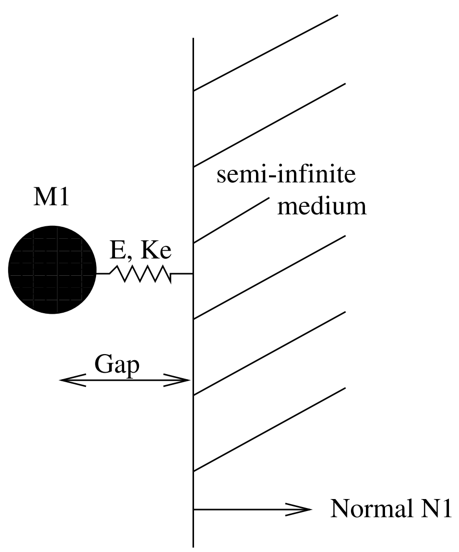
The computation restart is also checked.

Goal Get the evolution of the impact force according to the displacement and get the evolution of the plastic displacement according to the time.

Reference

Version 97' customer version

Model description



$$\begin{aligned} M1 &= 100000kg \\ Gap &= 0 \\ E &= 50Pa \\ K_e &= 100N/m \end{aligned}$$

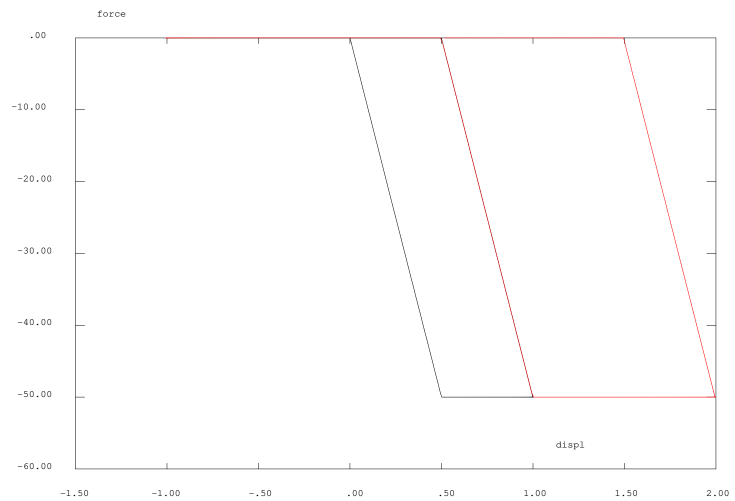
Test dy_devo6 **Results**

Results

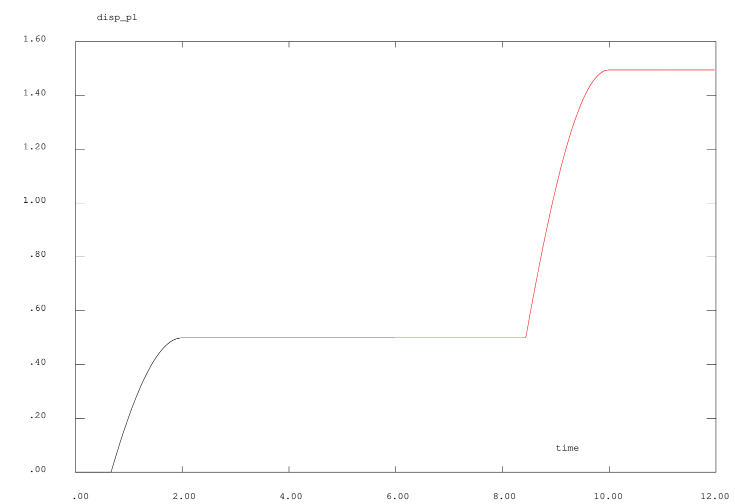
Maxima of plastic displacements :

	Analytic	CASTEM	Margin
1st computation	0.5 m	0.4993 m	0.14 %
2nd computation	1.5 m	1.4947 m	0.35 %

CASTEM figures



force - displacement



plastic displacement - time

```
*        Test Dy_devo6.dgibi: Jeux de données       *
*        -----------------------------------        *
*                                                   *
*****************************************************
*
*          DYNE       LIAISON POIN_PLAN
*      avec plastification du ressort de choc
*
*    il s'agit d'une grande masse animée d'un mouvement
*    sinusoidale (avec deux amplitudes differentes) qui
*    impacte un plan
*
*    on teste aussi la reprise de calcul
*
*****************************************************
opti echo 0 ;
graph ='N'  ;
*****************************************************
m1 = 100000 ;    comm 'masse impactée';
ke = 100. ;      comm 'raideur de choc';
se1 = 50. ;      comm 'limite élastique';
dt = .01 ;       comm 'pas de temps';
t1 = 12 ;        comm 'durée du calcul';
*****************************************************
opti dime 3 ;
pp1 = 0 0 0 ;
p1 = 0 0 0 ;
*
tab1 = table 'BASE_MODALE' ;
TAB2 = TABLE 'BASE_DE_MODES' ;
TAB1 . MODES = TAB2 ;
TAB2 . 1 = TABLE 'MODE' ;
TAB2 . 1 .'POINT_REPERE' = PP1 ;
TAB2 . 1 . 'FREQUENCE' = 0 ;
TAB2 . 1 .'MASSE_GENERALISEE' = M1 ;
TAB2 . 1 .'DEFORMEE_MODALE' = manu chpo p1 1 ux 1. ;
*
** definition du chargement
w0 = .125*2*pi ;
list1 = prog 0. pas dt (t1/2-dt) ;
list2 = prog (t1/2) pas dt t1 ;
lisy1 = prog sinu .125 ampl (w0**2) list1 ;
lisy2 = prog sinu .125 ampl (1.5*(w0**2)) list2 ;
ev1 = evol manu 'temps' (list1 et list2)
         'accel' (lisy1 et lisy2) ;
fx1 = force fx (-1*m1) p1 ;
cha1 = char fx1 ev1 ;
*
** chargement et cond. initiales en base A
cham1 = pjba tab1 cha1 ;
TCHA1 = TABLE 'CHARGEMENT' ;
TCHA1.'BASE_A' = CHAM1  ;
TINIT = TABLE 'INITIAL' ;
TINIT.'VITESSE' = manu chpo pp1 1 alfa w0 ;
*
** definition de la liaison
TLIA = TABLE 'LIAISON' ;
TLB = TABLE 'LIAISON_B' ;
TLIA.'LIAISON_B' = TLB ;
TL1 = TABLE 'LIAISON_ELEMENTAIRE' ;
TL1.'TYPE_LIAISON' = MOT 'POINT_PLAN';
TL1.'SUPPORT' = P1 ;
TL1.'NORMALE' = (1 0 0);
TL1.'RAIDEUR' = KE ;
TL1.'JEU' = 0.;
tl1.'SEUIL_PLASTIQUE' = se1 ;
TL1.'AMORTISSEMENT'= 0. ;
TLB. 1 = TL1 ;
*
** definition de la sortie
TSORT = TABLE 'SORTIE' ;
TSORL = TABLE 'LIAISON_B' ;
TSORT.'LIAISON_B' = TSORL ;
TSORL.TL1 = VRAI ;
*
** nombre de pas
npas = (enti (t1/dt)) ;
npas = npas / 2 ;
nins = 1 ;
*
** 1er bloc de calcul
tres1 = dyne differences_centrees tlia tab1 tcha1
          tsort tinit npas dt nins;
TTL1 = TRES1.TL1 ;
LIS_TEM0 = TRES1.'TEMPS_DE_SORTIE';
lisxp1 = TTL1.'UX' ;
lisyp1 = ttl1 .'FORCE_DE_CHOC' ;
titre 'force - deplacement';
evdf1 = evol manu 'depl' lisxp1 'force' lisyp1 ;
titre 'deplacement plastique - temps' ;
evdp1 = evol manu 'temps' lis_tem0 'dep_pl'
                          ttl1.deplacement_plastique ;
*
** reprise de calcul
tinit = table initial ;
tinit.reprise = tres1.reprise ;
tres2 = dyne de_vogelaere tlia tab1 tcha1 tsort
        tinit npas dt nins ;
TTL2 = TRES2.TL1 ;
LIS_TEM2 = TRES2.'TEMPS_DE_SORTIE';
lisxp2 = TTL2.'UX' ;
lisyp2 = ttl2 .'FORCE_DE_CHOC' ;
titre 'force - deplacement';
evdf2 = evol rouge  manu 'depl' lisxp2 'force' lisyp2 ;
titre 'deplacement plastique - temps' ;
evdp2 = evol rouge manu 'temps' lis_tem2 'dep_pl'
                          ttl2.deplacement_plastique;
*
  si (neg graph  'N');
    titre 'force - deplacement';
    dess (evdf1  et evdf2) ;
    titre 'deplacement plastique - temps' ;
    dess (evdp1 et evdp2) ;
  finsi;
*
* maxima des  deplacements  plastiques theoriques
d11 = .5 ; d22 = 1.5 ;
mess
'max des depl.  plastiques theoriques'd11'     ' d22;
d1 = maxi (extr evdp1 ordo) ;
d2 = maxi (extr evdp2 ordo) ;
mess
'max des depl.  plastiques  C2000    'd1'     ' d2 ;
er1 = abs(d1/d11  - 1); er2 = abs (d2/d22 - 1);
mess
'erreur                              'er1'%     'er2'%';
err =  maxi (prog  er1 er2) ;
 si (err > .005) ;
   erre  5 ;
 sinon ;
   erre  0 ;
 finsi;

fin ;
```

Test dy_devo6 **Comments**

1. **Definition of initial conditions**

   TINIT = TABLE 'INITIAL' ;
   TINIT.'VITESSE' = manu chpo pp1 1 alfa w0 ;

   For this first block of the computation initial values must be given in TINIT table of INITIAL subtype. The load is defined on A basis, then the component of the initial values is alfa (A basis is related to uncoupled equations).

2. **Definition of the linkage**

   TLIA = TABLE 'LIAISON' ;
   TLB = TABLE 'LIAISON_B' ;
   TLIA.'LIAISON_B' = TLB ;
   TL1 = TABLE 'LIAISON_ELEMENTAIRE' ;
   TL1.'TYPE_LIAISON' = MOT 'POINT_PLAN';
   TL1.'SUPPORT' = P1 ;
   TL1.'NORMALE' = (1 0 0);
   TL1.'RAIDEUR' = KE ;
   TL1.'JEU' = 0.;
   tl1.'SEUIL_PLASTIQUE' = se1 ;
   TL1.'AMORTISSEMENT'= 0. ;
   TLB. 1 = TL1 ;

   TLIA is a table of subtype LIAISON.
   TLIA.'LIAISON_B' (or TLB) is a table which gathers all linkages on B basis. This table has as many indexes as the number of linkages.
   TLB.1 (or TL1) defines an elementary linkage. It is a table of 'LIAISON_ELEMENTAIRE' subtype. The first index to be specified is the type of the linkage (here 'POINT_PLAN').
   The POINT-PLAN linkage allows to model the impact of a point onto a semi-infinite medium which is delimited by a fixed plane. The linkage is given in form of an imaginary spring which is defined by a stiffness (TL1.'RAIDEUR'), a yield limit (tl1.'SEUIL_PLASTIQUE') in case of plastification and a damping (TL1.'AMORTISSEMENT') in case of damping.
   Other data is related to the impacting point (TL1.'SUPPORT'), the initial gap between the point and the fixed plane (TL1.'JEU'), the normal perpendicular to the plane and oriented towards the inside of the solid (TL1.'NORMALE').

3. **Declaration of the output table**

   ** definition of output
   TSORT = TABLE 'SORTIE' ;
   TSORL = TABLE 'LIAISON_B' ;
   TSORT.'LIAISON_B' = TSORL ;
   TSORL.TL1 = VRAI ;

   TSORT is the table of SORTIE subtype.
   TSORT.'LIAISON_B' (or TSORL) is a table of LIAISON_B subtype and defines all linkages on B basis to be saved.
   Every index of TSORL is the table defining the linkage to be saved (here TL1). The value of TSORL.TL1 is VRAI (true); thus all internal data of the linkage TL1 will be saved.

4. **Output of results**

TTL1 = TRES1.TL1 ;
LIS_TEM0 = TRES1.'TEMPS_DE_SORTIE';
lisxp1 = TTL1.'UX' ;
lisyp1 = ttl1 .'FORCE_DE_CHOC' ;
titre 'force - displacement';
evdf1 = evol manu 'displ' lisxp1 'force' lisyp1 ;
titre 'plastic displacement - time' ;
evdp1 = evol manu 'time' lis_tem0 'disp_pl'
ttl1.deplacement_plastique ;

TRES.TL1 (or TTL1) is a table which contains the results related to the linkage defined in TL1 table.
- TTL1.'UX' is the list of UX displacements.
- ttl1 .'FORCE_DE_CHOC' is the list of the force of impact.
- ttl1.deplacement_plastique is the list of the plastic displacements.

5. **Restart of the computation**

tinit = table initial ;
tinit.reprise = tres1.reprise ;
tres2 = dyne de_vogelaere tlia tab1 tcha1 tsort tinit npas dt nins ;

For this second block of the computation initial values are defined in tinit.reprise. All data are given in the table tres1.reprise.

12.11 Test dy_devo3 Description sheet

Test name dy_devo3

Calculation type MECHANICS DYNAMICS 3D
IMPACT, LINKAGE CIRCLE-PLANE-FRICTION

Topic
A wheel subjected to a torsional moment

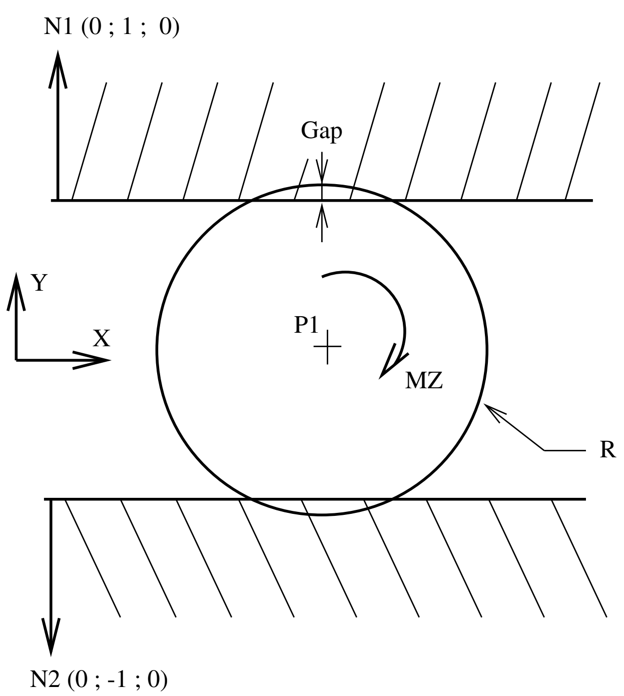
A wheel of 1 meter diameter is clamped by 2 planes. The wheel is subjected to a torsional moment.
The linkages used are a circle-point with taking into consideration the friction.

Goal Find the rotation around Z-axis.

Reference

Version 97' customer version

Model description



$$
\begin{array}{ll}
R & = 1m \\
Gap & = -0.00025m \\
M_z & = 1000N.m \\
K_e & = 4.10^6 N/m \\
K_t & = (10 * K_e)/R^2 \\
C & = 0N.s/m \\
C_t & = \sqrt{K_t * M} \\
Coef_{sliding} & = 0.4 \\
Coef_{adherence} & = 0.4
\end{array}
$$

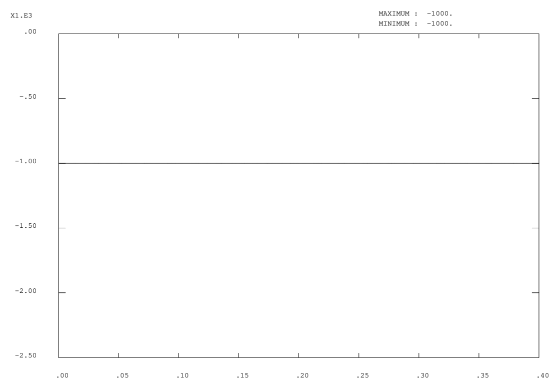
Test dy_devo3

Results

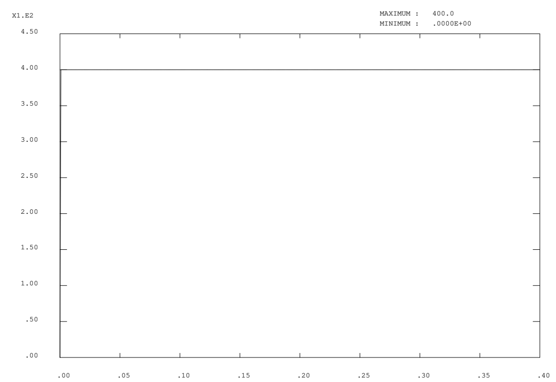
Results

Time	0.4
Rotation RZ	16.013

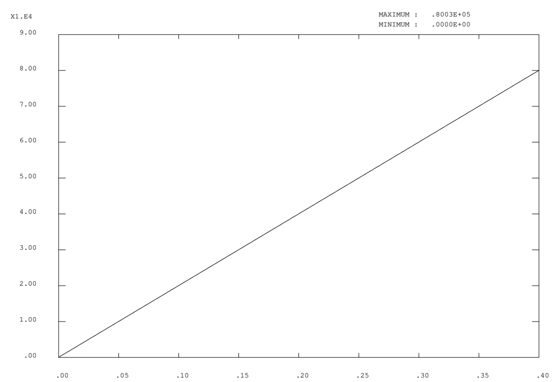
CASTEM figures



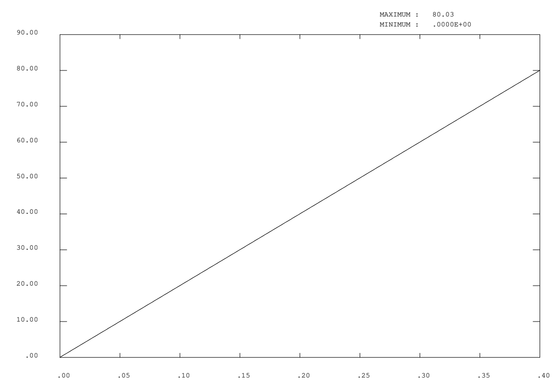
***** - TL1- NORMAL IMPACT FORCE *****



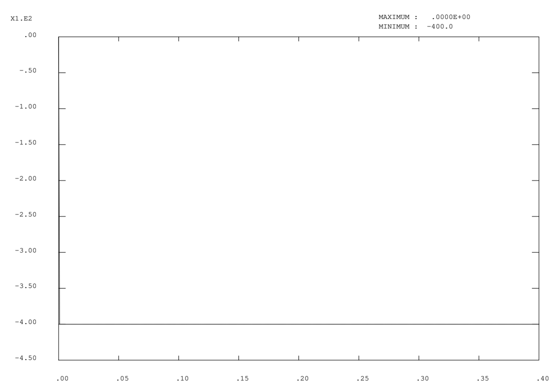
***** - TL1- TANGENTIAL IMPACT FORCE *****



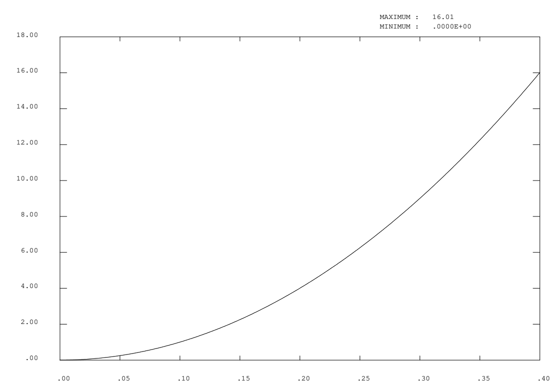
***** - TL1- WEAR POWER *****



***** - TL1- VX CONTACT POINT *****



***** - TL1- MZ POINT OF MEDIAN LINE *****



***** - TL1- RZ POINT OF MEDIAN LINE *****

```
*        Test Dy_devo3.dgibi: Jeux de données        *
*        ------------------------------------        *
*                                                    *
* si GRAPH = N, les graphiques ne sont pas affichés
* si GRAPH différent de N, tous les graphiques
* sont affichés
*
GRAPH = 'N' ;
*
SAUT PAGE ;
*----------------------------------------------------*
*                   DY_DEVO3                         *
*----------------------------------------------------*
*                                                    *
*    Tests de l'op{rateur DYNE option DE_VOGELAERE   *
*    ------------------------------------------      *
*                                                    *
*                                                    *
*   Une roue de diamètre 1 mètre bloquée par 2 plans  *
*   On applique à la roue un moment Mz               *
*   2 liaisons CERCLE_PLAN_FROTTEMENT                *
*                                                    *
*----------------------------------------------------*
*
OPTION ECHO 0 ;
TEMPS ;
OPTION ELEM SEG2 DIME 3 ;
*
*     Définition géométrie
*
P1 = 0. 0. 0. ;
*
*     Base modale
*
CHP_M1 =
       MANU CHPO 6 P1  'UX'  1.0  'UY'  0.0  'UZ'  0.0
                       'RX'  0.0  'RY'  0.0  'RZ'  0.0;
CHP_M2 =
       MANU CHPO 6 P1  'UX'  0.0  'UY'  1.0  'UZ'  0.0
                       'RX'  0.0  'RY'  0.0  'RZ'  0.0;
CHP_M3 =
       MANU CHPO 6 P1  'UX'  0.0  'UY'  0.0  'UZ'  0.0
                       'RX'  0.0  'RY'  0.0  'RZ'  1.0;
*
TBAS = TABLE 'BASE_MODALE' ;
TBAS . 'MODES' = TABLE 'BASE_DE_MODES' ;
TBAS . 'MODES' . 'MAILLAGE' = P1  ;
TBAS . 'MODES' . 1 = TABLE 'MODE' ;
TBAS . 'MODES' . 1 . 'POINT_REPERE'      = ( 0 0 0 ) ;
TBAS . 'MODES' . 1 . 'FREQUENCE'         =   0.0     ;
TBAS . 'MODES' . 1 . 'MASSE_GENERALISEE' =   1.0     ;
TBAS . 'MODES' . 1 . 'DEFORMEE_MODALE'   =   CHP_M1  ;
TBAS . 'MODES' . 2 = TABLE 'MODE' ;
TBAS . 'MODES' . 2 . 'POINT_REPERE'      = ( 0 0 0 ) ;
TBAS . 'MODES' . 2 . 'FREQUENCE'         =   0.0     ;
TBAS . 'MODES' . 2 . 'MASSE_GENERALISEE' =   1.0     ;
TBAS . 'MODES' . 2 . 'DEFORMEE_MODALE'   =   CHP_M2  ;
TBAS . 'MODES' . 3 = TABLE 'MODE' ;
TBAS . 'MODES' . 3 . 'POINT_REPERE'      = ( 0 0 0 ) ;
TBAS . 'MODES' . 3 . 'FREQUENCE'         =   0.0     ;
TBAS . 'MODES' . 3 . 'MASSE_GENERALISEE' =   1.0     ;
TBAS . 'MODES' . 3 . 'DEFORMEE_MODALE'   =   CHP_M3  ;
*
*     Champ de force
*
XFOR   =  1000.0  ;
CHP_F1 = MANU CHPO 3 P1  'MX'  0.0
                         'MY'  0.0
                         'MZ'  XFOR ;
EVO_F1 = EVOL MANU ( PROG -1. 10. ) ( PROG 1. 1. ) ;
CHA_F1 = CHARGE 'FORC' CHP_F1 EVO_F1 ;
TCHARG = TABLE 'CHARGEMENT' ;
TCHARG . 'BASE_A' = PJBA TBAS CHA_F1 ;
*
*---  Définition des liaisons
*
TLIA1 = TABLE 'LIAISON' ;
TLB1  = TABLE 'LIAISON_B' ;
TLIA1 . 'LIAISON_B' = TLB1 ;
*
*   Liaison CERCLE_PLAN_FROTTEMENT avec amortissement
*
RAY_SUP = 1.0   ;
XNU    =  0.4   ;
XRAID  =  4.0E6 ;
RAID_T =  ( 10.0 * XRAID ) / ( RAY_SUP ** 2 )  ;
AMOR_T =  ( RAID_T * 1.0 ) ** 0.5 ;
TL1  =  TABLE  'LIAISON_ELEMENTAIRE'      ;
TL1.'TYPE_LIAISON'             =
     MOT 'CERCLE_PLAN_FROTTEMENT'                        ;
TL1.'SUPPORT'                  =  P1                     ;
TL1.'JEU'                      =  -0.00025               ;
TL1.'NORMALE'                  =  ( 0 -1  0 )            ;
TL1.'RAIDEUR'                  =  XRAID                  ;
TL1.'RAYON_SUPPORT'            =  RAY_SUP                ;
TL1.'COEFFICIENT_GLISSEMENT'   =  XNU                    ;
TL1.'COEFFICIENT_ADHERENCE'    =  XNU                    ;
TL1.'RAIDEUR_TANGENTIELLE'     =  RAID_T                 ;
TL1.'AMORTISSEMENT_TANGENTIEL' =  AMOR_T                 ;
TL1.'AMORTISSEMENT'            =  0.0                    ;
TL2  =  TABLE  'LIAISON_ELEMENTAIRE'                     ;
TL2.'TYPE_LIAISON'             =
     MOT 'CERCLE_PLAN_FROTTEMENT'                        ;
TL2.'SUPPORT'                  =  P1                     ;
TL2.'JEU'                      =  -0.00025               ;
TL2.'NORMALE'                  =  ( 0  1  0 )            ;
TL2.'RAIDEUR'                  =  XRAID                  ;
TL2.'RAYON_SUPPORT'            =  RAY_SUP                ;
TL2.'COEFFICIENT_GLISSEMENT'   =  XNU                    ;
TL2.'COEFFICIENT_ADHERENCE'    =  XNU                    ;
TL2.'RAIDEUR_TANGENTIELLE'     =  RAID_T                 ;
TL2.'AMORTISSEMENT_TANGENTIEL' =  AMOR_T                 ;
TL2.'AMORTISSEMENT'            =  0.0                    ;
*
TLB1 . 1  = TL1  ;
TLB1 . 2  = TL2  ;
*
*---  Définition des résultats en sortie
*
TSORT = TABLE 'SORTIE' ;
TSOR1 = TABLE 'SORTIE' ;
TSORT . 'VARIABLE'    = TSOR1 ;
TSOR1 . 'DEPLACEMENT' =  FAUX ;
TSOR1 . 'VITESSE'     =  FAUX ;
*
TSOR2 = TABLE 'SORTIE' ;
TSORT . 'LIAISON_B' = TSOR2 ;
TSOR2 . TL1  = VRAI ;
*
*     Temps
*
PDT  = 4.0E-5 ;
NP1  =  5000  ;
NP2  =  5000  ;
NINS =    20  ;
*
```

```
*------  Opérateur DYNE  ------
*
TRESU1 = DYNE 'DIFFERENCES_CENTREES' TBAS TCHARG TSORT
                                   TLIA1 NP1 PDT NINS ;
*
TINIT = TABLE 'INITIAL' ;
TINIT.'REPRISE' = TRESU1.'REPRISE' ;
TRESU2 = DYNE 'DE_VOGELAERE' TBAS  TCHARG  TSORT  TLIA1
                                   TINIT NP2 PDT NINS ;
*
*     Tracé des courbes
*
LISTE1 = TRESU1.'TEMPS_DE_SORTIE' ;
LISTE2 = TRESU2.'TEMPS_DE_SORTIE' ;
*
TITRE '***** - TL1-  FORCE DE CHOC NORMALE  *****' ;
EVO_FN1 =
EVOL MANU LISTE1  TRESU1.TL1.'FORCE_DE_CHOC_NORMALE'  ;
EVO_FN2 =
EVOL MANU LISTE2  TRESU2.TL1.'FORCE_DE_CHOC_NORMALE'  ;
EVO_FN = CONCAT EVO_FN1 EVO_FN2 ;
*
TITRE
'***** - TL1-  FORCE DE CHOC TANGENTIELLE  *****' ;
EVO_FT1 = EVOL MANU LISTE1
          TRESU1.TL1.'FORCE_DE_CHOC_TANGENTIELLE' ;
EVO_FT2 = EVOL MANU LISTE2
          TRESU2.TL1.'FORCE_DE_CHOC_TANGENTIELLE' ;
EVO_FT = CONCAT EVO_FT1 EVO_FT2 ;
*
TITRE '***** - TL1-  PUISSANCE USURE *****' ;
EVO_PU1 = EVOL MANU LISTE1
          TRESU1.TL1.'PUISSANCE_USURE_INSTANTANEE' ;
EVO_PU2 = EVOL MANU LISTE2
          TRESU2.TL1.'PUISSANCE_USURE_INSTANTANEE' ;
EVO_PU = CONCAT EVO_PU1 EVO_PU2 ;
*
TITRE '***** - TL1-  VX POINT DE CONTACT  *****' ;
EVO_VX1 = EVOL MANU LISTE1
          TRESU1.TL1.'VX_POINT_CONTACT' ;
EVO_VX2 = EVOL MANU LISTE2
          TRESU2.TL1.'VX_POINT_CONTACT' ;
EVO_VX = CONCAT EVO_VX1 EVO_VX2 ;
*
TITRE '***** - TL1-  MZ POINT FIBRE NEUTRE  *****' ;
EVO_MZ1 = EVOL MANU LISTE1
          TRESU1.TL1.'MZ_POINT_FIBRE_NEUTRE' ;
EVO_MZ2 = EVOL MANU LISTE2
          TRESU2.TL1.'MZ_POINT_FIBRE_NEUTRE' ;
EVO_MZ = CONCAT EVO_MZ1 EVO_MZ2 ;
*
TITRE '***** - TL1-  RZ POINT FIBRE NEUTRE  *****' ;
EVO_RZ1 = EVOL MANU LISTE1
          TRESU1.TL1.'RZ_POINT_FIBRE_NEUTRE' ;
EVO_RZ2 = EVOL MANU LISTE2
          TRESU2.TL1.'RZ_POINT_FIBRE_NEUTRE' ;
EVO_RZ = CONCAT EVO_RZ1 EVO_RZ2 ;
*
SI ( NEG GRAPH 'N' ) ;
   DESS EVO_FN 'MIMA' ;
   DESS EVO_FT 'MIMA' ;
   DESS EVO_PU 'MIMA' ;
   DESS EVO_VX 'MIMA' ;
   DESS EVO_MZ 'MIMA' ;
   DESS EVO_RZ 'MIMA' ;
FINSI ;
*
LIS_RZ = EXTRAIRE EVO_RZ 'ORDO' ;
LISTEM = EXTRAIRE EVO_RZ 'ABSC' ;
D = EXTRAIRE  LIS_RZ  501 ;
T = EXTRAIRE  LISTEM  501 ;
SAUT PAGE ;
MESS                    'valeur du temps :' T ;
MESS 'valeur de la rotation autour de Z :' D ;
SAUT 2 LIGNES ;
*
TEMPS ;
*
*     Code de bon fonctionnement
*
DEPREF = 100. * T * T ;
RESI = ABS ( ( D - DEPREF ) / DEPREF ) ;
SI ( RESI <EG 1E-2 ) ;
   ERRE 0 ;
SINON ;
   ERRE 5 ;
FINSI ;
*
FIN ;
```

Test dy_devo3 **Comments**

1. **Linkage CERCLE_PLAN_FROTTEMENT**

   TL1 = TABLE 'LIAISON_ELEMENTAIRE' ;
   TL1.'TYPE_LIAISON' = MOT 'CERCLE_PLAN_FROTTEMENT' ;
   TL1.'SUPPORT' = P1 ;
   TL1.'JEU' = -0.00025 ;
   TL1.'NORMALE' = (0 -1 0) ;
   TL1.'RAIDEUR' = XRAID ;
   TL1.'RAYON_SUPPORT' = RAY_SUP ;
   TL1.'COEFFICIENT_GLISSEMENT' = XNU ;
   TL1.'COEFFICIENT_ADHERENCE' = XNU ;
   TL1.'RAIDEUR_TANGENTIELLE' = RAID_T ;
   TL1.'AMORTISSEMENT_TANGENTIEL' = AMOR_T ;
   TL1.'AMORTISSEMENT' = 0.0 ;

   The linkage CERCLE_PLAN_FROTTEMENT is used for impacts of a circle onto a semi-infinite medium. The circle is simply described by its center point (TL1.'SUPPORT') and its radius (TL1.'RAYON_SUPPORT').
   TL1.'JEU' is the gap between a point of the circle and the medium. Here the gap is negative : it means the medium penetrates into the circle.
   The linkage takes into consideration the friction, so specific data must be given to describe the imaginary spring : the impact stiffness (TL1.'RAIDEUR'), the tangential stiffness (TL1.'RAIDEUR_TANGENTIELLE'), the sliding coefficient (TL1.'COEFFICIENT_GLISSEMENT'), the adherence stress coefficient (TL1.'COEFFICIENT_ADHERENCE'), the impact damping (TL1.'AMORTISSEMENT') and the tangential damping (TL1.'AMORTISSEMENT_TANGENTIEL').
   Note that the impact damping is optional.

2. **Use of 2 linkages**

   TL1 = TABLE 'LIAISON_ELEMENTAIRE' ;
   TL1.'TYPE_LIAISON' = MOT 'CERCLE_PLAN_FROTTEMENT' ;
   TL1.'SUPPORT' = P1 ;
   ...
   TL1.'NORMALE' = (0 -1 0) ;
   ...
   TL2 = TABLE 'LIAISON_ELEMENTAIRE' ;
   TL2.'TYPE_LIAISON' = MOT 'CERCLE_PLAN_FROTTEMENT' ;
   TL2.'SUPPORT' = P1 ;
   ...
   TL2.'NORMALE' = (0 1 0) ;

   The difference between TL1 and TL2 is only made by the definition of the normals TL1.'NORMALE' and TL2.'NORMALE'.

3. **Declaration of the output table**

```
TSORT = TABLE 'SORTIE' ;
TSOR1 = TABLE 'SORTIE' ;
TSORT . 'VARIABLE' = TSOR1 ;
TSOR1 . 'DEPLACEMENT' = FAUX ;
TSOR1 . 'VITESSE' = FAUX ;
*
TSOR2 = TABLE 'SORTIE' ;
TSORT . 'LIAISON_B' = TSOR2 ;
TSOR2 . TL1 = VRAI ;
```

The output table will contain only the results related to TL1 linkage.

4. **Output of results**

```
LISTE1 = TRESU1.'TEMPS_DE_SORTIE' ;
LISTE2 = TRESU2.'TEMPS_DE_SORTIE' ;
...
TITRE '***** - TL1- RZ POINT OF MEDIAN LINE *****' ;
EVO_RZ1 = EVOL MANU LISTE1 TRESU1.TL1.'RZ_POINT_FIBRE_NEUTRE' ;
EVO_RZ2 = EVOL MANU LISTE2 TRESU2.TL1.'RZ_POINT_FIBRE_NEUTRE' ;
EVO_RZ = CONCAT EVO_RZ1 EVO_RZ2 ;
...
LIS_RZ = EXTRAIRE EVO_RZ 'ORDO' ;
LISTEM = EXTRAIRE EVO_RZ 'ABSC' ;
D = EXTRAIRE LIS_RZ 501 ;
T = EXTRAIRE LISTEM 501 ;
```

The computation is done in two parts: thus the results are separate (some in TRESU1 and others in TRESU2). The CONCAT operator performs the joining up of two objects (of EVOLUTION type). To obtain a specific value from the EVO_RZ object, it is first necessary to extract every list of reals from the EVOLUTION object (EVO_RZ) : the EXTRAIRE operator extracts the lists of reals related to the abscissas ('ABSC') and the ordinates ('ORDO'). Then you can extract the required value from the obtained list of reals : 501 is the row of the required value in the list.

12.12 Test dy_devo4 Description sheet

Test name dy_devo4

Calculation type MECHANICS DYNAMICS 3D
IMPACT, LINKAGE SECTION-SECTION-OUTSIDE

Finite element type POUT

Topic
Impact of a beam with concentrated mass against a plane.

A concentrated mass is fixed at the extremity of a cantilever beam. A normal force is applied on this tips, this load gives the initial displacements of the beam.

Two different linkages are used in this example : the PROFIL_PROFIL_SECTION linkage is tested then compared with the POINT_PLAN linkage.

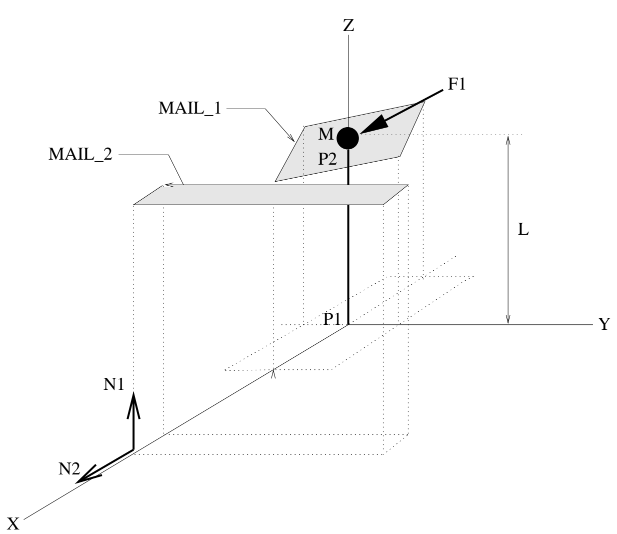
Goal Find the impact force and displacement UX at time T = 0.4 s.

Reference

Version 97' customer version

Model description

L	$= 0.9m$	E	$= 2.1.10^{11}Pa$
M	$= 2kg$	ν	$= 0.3$
A	$= 1.1.10^{-6}m^2$	ρ	$= 78kg/m^3$
I_y	$= 8.33.10^{-10}m^4$	$F1$	$= 1N$
I_z	$= 8.33.10^{-10}m^4$	K_e	$= 1.10^5 N/m$
G	$= 5.10^{-7}m^4$	Gap	$= 10^{-3}m$

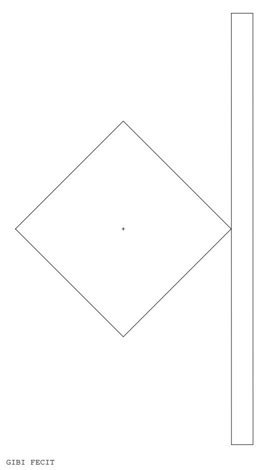


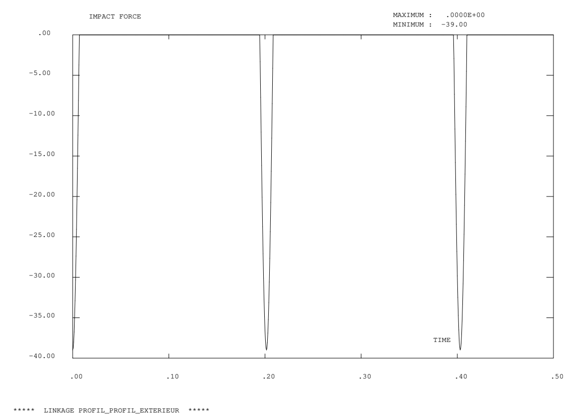
Test dy_devo4 **Results**

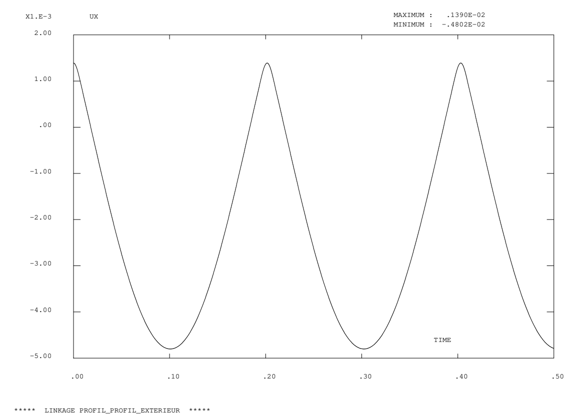
Results (t=0.4s)

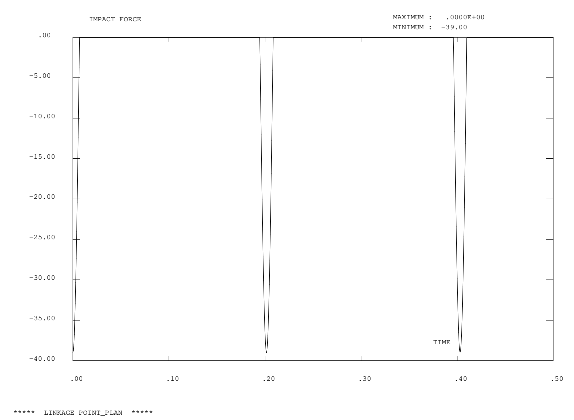
	PROFIL_PROFIL_EXTERIEUR	POINT_PLAN
Impact force	-29.833 N	-29.833 N
Displacement UX	$1.298.10^{-3}$ m	$1.298.10^{-3}$ m

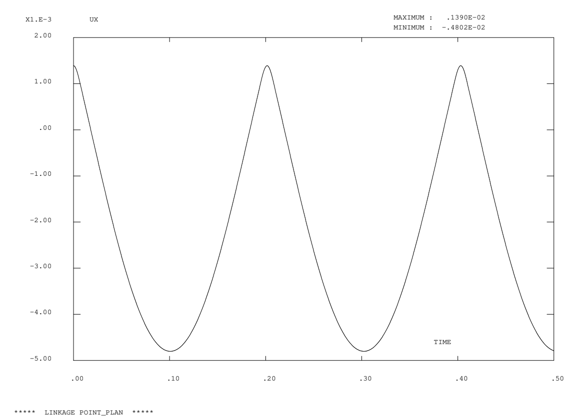
CASTEM figures











```
*        Test Dy_devo4.dgibi: Jeux de données       *
*        -----------------------------------        *
*                                                   *
* si GRAPH = N, les graphiques ne sont pas affichés
* si GRAPH différent de N, tous les graphiques
* sont affichés
*
GRAPH = 'N' ;
*
SAUT PAGE ;
*
*---------------------------------------------------*
*                   DY_DEVO4                        *
*---------------------------------------------------*
*                                                   *
*    Tests de l'op{rateur DYNE option DE_VOGELAERE   *
*    --------------------------------------------   *
*                                                   *
*                                                   *
* 1 poutre encastrée, 10 éléments finis,            *
* masse négligeable                                 *
* 1 masse concentrée à son extrémité                *
* Test PROFIL_PROFIL_EXTERIEUR avec des carrés      *
* Comparaison avec la liaison POINT_PLAN            *
* NB_M: nombre de segments sur le coté du carré mobile*
* NB_F: nombre de segments sur le coté du carré fixe  *
*                                                   *
*---------------------------------------------------*

OPTION ECHO 0 ;
TEMPS ;
*
NB_M = 3  ;
NB_F = 1  ;
R_PPL = 1.E+05 ;
R_PP  = R_PPL  ;
EXPO = 0.5 ;
*
OPTION ELEM SEG2 DIME 3 ;
*
*     Définition géométrie
*
P1 = 0. 0. 0. ;
P2 = 0. 0. 0.9 ;
LIG1 = P1 D 10 P2 ;
*
*     Définition matériau
*
MODL1 = MODL LIG1 MECANIQUE ELASTIQUE POUT;

MA1 = MATR MODL1 YOUNG 2.1E11 NU 0.3 RHO 78.  ;
CA1 = CARB MODL1 SECT 1.1E-6  INRY 8.33E-10
           INRZ 8.33E-10 TORS 5.0E-7  VECT (1. 0. 1. );
*
RIG1 = RIGI MODL1 (MA1 ET CA1) ;
MAST = (MASSE MODL1  (MA1 ET CA1)) ET
       ( MASSE DEPL 2.0 P2 ) ;
*
*     Conditions aux limites
*
ENC1 = BLOQUE DEPL ROTA P1 ;
ENC2 = BLOQUE 'UY' LIG1 ;
RIGT = RIG1 ET ENC1 ET ENC2 ;
*
*     Calcul des modes
*
MOD1=VIBRATION PROCHE ( PROG 1.0  )  RIGT MAST ;
*
*     Base modale
*
TB1 = TRADUIRE MOD1 ;
TBAS = TABLE 'BASE_MODALE' ;
TBAS.'MODES' = TB1 ;
*
RIGI1 = RIGI TB1 ;
*
*     Amortissement
*
AMOR1 = AMOR TBAS ( PROG 0.0 ) ;
*
TA = TABLE 'AMORTISSEMENT' ;
TA.'AMORTISSEMENT' = AMOR1 ;
*
*     Déplacement initial
*
FE2 = FORCE FX 1. P2 ;
XE2 = RESOU RIGT  FE2   ;
XMAX = MAXI XE2 AVEC (MOTS UX ) ;
FE1 = FE2 * (0.139E-2 / XMAX );
FGE1 = PJBA TBAS FE1 ;
ALO = RESO RIGI1 FGE1 ;
*
TINIT = TABLE 'INITIAL' ;
TINIT.'DEPLACEMENT' = ALO ;
*
*---  Définition des liaisons
*
TLIA = TABLE 'LIAISON' ;
TLB = TABLE 'LIAISON_B' ;
TLIA.'LIAISON_B' = TLB ;
NORM1 = 0 0 1 ;
*
*   Liaison PROFIL_PROFIL_EXTERIEUR
*
*   profil mobile
P1_1 =   -0.5    0.0   0.9 ;
P1_2 =    0.0   -0.5   0.9 ;
P1_3 =    0.5    0.0   0.9 ;
P1_4 =    0.0    0.5   0.9 ;
MAIL_1 = P1_1 D NB_M P1_2 D NB_M P1_3 D NB_M P1_4 D
         NB_M P1_1 ;
*
*   profil fixe
P2_1 =    0.501   1.0    0.9 ;
P2_2 =    0.501  -1.0    0.9 ;
P2_3 =    0.601  -1.0    0.9 ;
P2_4 =    0.601   1.0    0.9 ;
MAIL_2 = P2_1 D NB_F P2_2 D NB_F P2_3 D NB_F P2_4 D
         NB_F P2_1 ;
*
SI ( NEG GRAPH 'N' ) ;
   TRAC ( 0 0 1000 ) ( MAIL_1 ET MAIL_2 ) ;
FINSI ;
*
TL1  =  TABLE  'LIAISON_ELEMENTAIRE'    ;
TL1.'TYPE_LIAISON'     = MOT 'PROFIL_PROFIL_EXTERIEUR';
TL1.'SUPPORT'          =      P2     ;
TL1.'PROFIL_FIXE'      =    MAIL_2   ;
TL1.'PROFIL_MOBILE'    =    MAIL_1   ;
TL1.'NORMALE'          =    NORM1    ;
TL1.'RAIDEUR'          =    R_PP     ;
TL1.'EXPOSANT_RAIDEUR' =    EXPO     ;
*
TLB . 1  = TL1  ;
*
*---  Définition des résultats en sortie
```

```
*
TSORT = TABLE 'SORTIE' ;
TSOR1 = TABLE 'SORTIE' ;
TSORT . 'VARIABLE'    = TSOR1 ;
TSOR1 . 'DEPLACEMENT' =  FAUX ;
TSOR1 . 'VITESSE'     =  FAUX ;
*
TSOR2 = TABLE 'SORTIE' ;
TSORT.'LIAISON_B' = TSOR2 ;
TSOR2.TL1  = VRAI ;
*
*     Temps
*
PDT  = 0.1E-3 ;
NP1  =  5000  ;
NINS =     5  ;
*
*------  Op{rateur DYNE  ------
*
TRESU1 = DYNE 'DE_VOGELAERE' TBAS TA TINIT TSORT TLIA
         NP1 PDT NINS ;
*
*     Trac{ des courbes
*
SI ( NEG GRAPH 'N' ) ;
 TITRE '*****  LIAISON PROFIL_PROFIL_EXTERIEUR  *****';
 DESS ( EVOL MANU 'TEMPS' TRESU1 . 'TEMPS_DE_SORTIE'
                  'FN'  TRESU1 . TL1 . 'FORCE_DE_CHOC')
                  'MIMA' ;
*
 DESS ( EVOL MANU 'TEMPS' TRESU1 . 'TEMPS_DE_SORTIE'
                  'UX'   TRESU1 . TL1 . 'UX' )  'MIMA';
FINSI ;
D_FC_1 = EXTRAIRE ( TRESU1 . TL1 . 'FORCE_DE_CHOC' )
         801 ;
D_DE_1 = EXTRAIRE ( TRESU1 . TL1 . 'UX' ) 801 ;
*
SAUT PAGE ;
MESS '-----> liaison PROFIL_PROFIL_EXTERIEUR' ;
MESS '           valeur de la force de choc :' D_FC_1 ;
MESS '                valeur du déplacement :' D_DE_1 ;
*
*   Liaison POINT_PLAN
*
NORM1 = 1  0  0 ;
TL1  =  TABLE  'LIAISON_ELEMENTAIRE'    ;
TL1.'TYPE_LIAISON'     =  MOT 'POINT_PLAN' ;
TL1.'SUPPORT'          =      P2     ;
TL1.'NORMALE'          =    NORM1    ;
TL1.'RAIDEUR'          =    R_PPL    ;
TL1.'JEU'              =    0.001    ;
*
TLB . 1  = TL1  ;
TSOR2.TL1  = VRAI ;
*
TRESU2 = DYNE 'DE_VOGELAERE' TBAS TA TINIT TSORT TLIA
         NP1 PDT NINS ;
*
*     Tracé des courbes
*
SI ( NEG GRAPH 'N' ) ;
   TITRE '*****  LIAISON POINT_PLAN  *****' ;
   DESS ( EVOL MANU 'TEMPS' TRESU2 . 'TEMPS_DE_SORTIE'
       'FN'    TRESU2 . TL1 . 'FORCE_DE_CHOC') 'MIMA' ;
*
   DESS ( EVOL MANU 'TEMPS' TRESU2 . 'TEMPS_DE_SORTIE'
       'UX'    TRESU2 . TL1 . 'UX' )  'MIMA' ;
FINSI ;
*
TEMPS ;
*
*     Code de bon fonctionnement
*
D_FC_2 =EXTRAIRE ( TRESU2 . TL1 . 'FORCE_DE_CHOC' ) 801;
D_DE_2 =EXTRAIRE ( TRESU2 . TL1 . 'UX' ) 801 ;
SAUT 2 LIGNES ;
MESS '-----> liaison POINT_PLAN ' ;
MESS '           valeur de la force de choc :' D_FC_2 ;
MESS '                valeur du déplacement :' D_DE_2 ;
SAUT 3 LIGNES ;
R_FC = ABS ( ( D_FC_1 - D_FC_2 ) / D_FC_2 ) ;
R_DE = ABS ( ( D_DE_1 - D_DE_2 ) / D_DE_2 ) ;
SI ( ( R_FC <EG 1E-2 ) OU ( R_DE <EG 1E-2 ) ) ;
   ERRE 0 ;
SINON ;
   ERRE 5 ;
FINSI ;
*
FIN ;
```

Test dy_devo4 **Comments**

1. **Creation of the table of BASE_MODALE subtype**

   TB1 = TRADUIRE MOD1 ;
   TBAS = TABLE 'BASE_MODALE' ;
   TBAS.'MODES' = TB1 ;

   The DYNE operator uses a BASE_MODALE subtype table but the VIBR operator creates a SOLUTION object if keyword TBAS is not specified. Also it is necessary to create a BASE_MODALE subtype table from a SOLUTION object of MODE subtype.
   First of all, the TRADUIRE operator creates a BASE_DE_MODES subtype table (TB1) from a SOLUTION object (MOD1). After the creation of a BASE_MODALE subtype table (TBAS), TB1 is given in TBAS.'MODES'.

2. **Generalized stiffnesses**

   RIGI1 = RIGI TB1 ;

   RIGI1 is the stiffness due to the modes contained in TB1 (BASE_DE_MODES subtype table) .

3. **Initial displacement**

   FGE1 = PJBA TBAS FE1 ;
   ALO = RESO RIGI1 FGE1 ;

   ALO is a CHPOINT object which has only one component (ALPHA) . Indeed this object was created with FGE1 load (of modal component) and RIGI1 (generalized stiffness).

4. **Linkage PROFIL_PROFIL_EXTERIEUR**

   TL1 = TABLE 'LIAISON_ELEMENTAIRE' ;
   TL1.'TYPE_LIAISON' = MOT 'PROFIL_PROFIL_EXTERIEUR' ;
   TL1.'SUPPORT' = P2 ;
   TL1.'PROFIL_FIXE' = MAIL_2 ;
   TL1.'PROFIL_MOBILE' = MAIL_1 ;
   TL1.'NORMALE' = NORM1 ;
   TL1.'RAIDEUR' = R_PP ;
   TL1.'EXPOSANT_RAIDEUR' = EXPO ;
   TLB . 1 = TL1 ;

   The linkage PROFIL_PROFIL_EXTERIEUR is used for impacts of a moving section onto a fixed section.
   The moving section (TL1.'PROFIL_MOBILE') is connected to the structure by a point (TL1.'SUPPORT') and is outside the fixed section (TL1.'PROFIL_FIXE'). The sections must be oriented anti clockwise and in rest position both the mesh sections must have any intersection point.
   The given normal (TL1.'NORMALE', N1 in the model description) must be perpendicular to the plane composed by both sections.
   The normal impact force F is given by : $F = -K * A^b$
   where K is the impact stiffness (TL1.'RAIDEUR'),
   A is the section surface which passed through fixed section,
   b is the stiffness exponent (TL1.'EXPOSANT_RAIDEUR').

5. **Linkage POINT_PLAN**

   To obtain the same configuration as the PROFIL_PROFIL_EXTERIEUR linkage, the value of the gap must be equal to 0.001 (the gap along X-axis between moving section and fixed section was of 0.001 too) and the normal must be oriented along X-axis (N2 in the model description).

12.13 Test dy_devo8 Description sheet

Test name dy_devo8

Calculation type MECHANICS DYNAMICS 3D
IMPACT
LINKAGE POINT-MOVING CIRCLE, LINKAGE LINE-LINE-FRICTION

Finite element type

Topic
Impact of a thrust against a moving circle.

A point impacts a moving circle. Two different linkages are used and compared in this test.

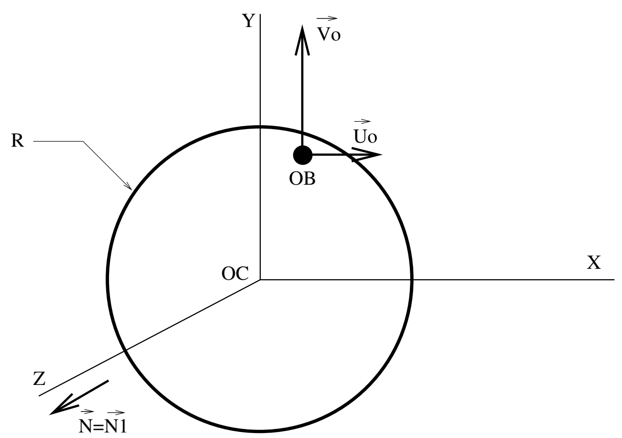
The POINT_CERCLE_MOBILE linkage gives the impact of a point onto a mobile circle, taking into consideration the dry friction in the tangent plane of contact. The LIGNE_LIGNE_FROTTEMENT gives the impact of a slave line against a master line.

Goal Find displacements UX and UY of the thrust. Comparison between both methods.

Reference

Version 97' customer version

Model description

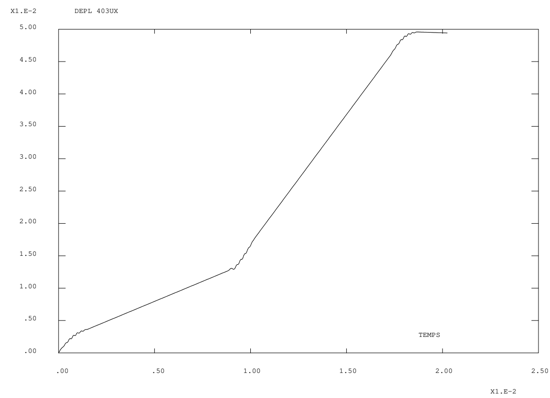


R	$= 0.04m$
U_o	$= 5m/s$
V_o	$= 10m/s$
K_e	$= 1.10^8 N/m$
K_t	$= -100N/m$
C_t	$= 0.01N.s/m$
$Coef_{sliding}$	$= 20$
$Coef_{adherence}$	$= 20$

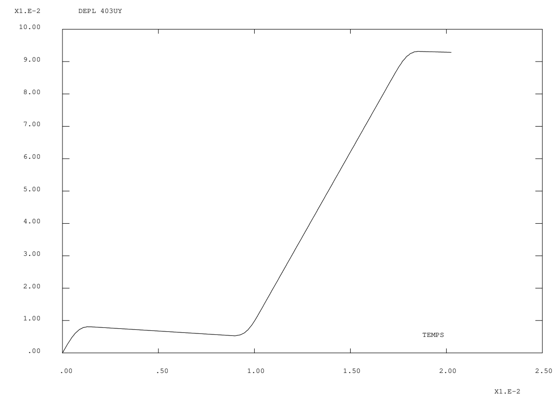
Test dy_devo8

Results

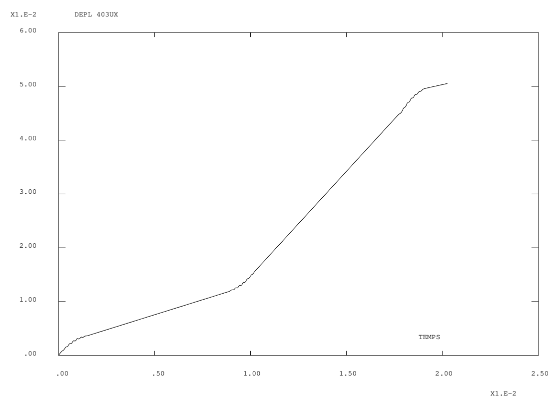
CASTEM figures



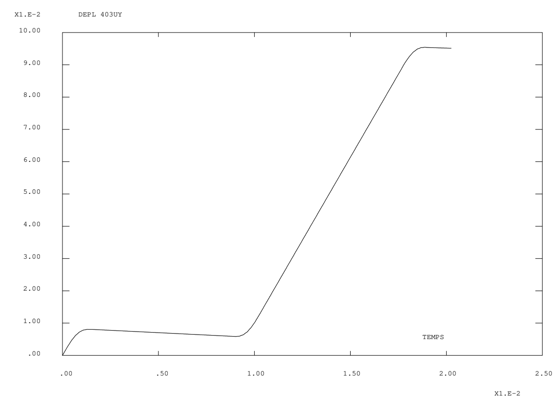
UX DISPLACEMENT OF OB : LIGNE_LIGNE_FROTTEMENT



UY DISPLACEMENT OF OB : LIGNE_LIGNE_FROTTEMENT



UX DISPLACEMENT OF OB : POINT_CERCLE_MOBILE



UY DISPLACEMENT OF OB : POINT_CERCLE_MOBILE

```
*       Test Dy_devo8.dgibi: Jeux de données
*       ------------------------------------
*
OPTION ECHO 0                   ;
OPTION ELEM SEG2 DIME 3         ;
********************************
*      PROBLÈME                 *
*   Test de Comparaison entre   *
* les liaisons                  *
* point_cercle_mobile et        *
* point_ligne .                 *
*                               *
*   Impact d'une butée          *
*  sur un cercle mobile .       *
*                               *
********************************
OC = 0. 0. 0.                   ;
A = 0.04 0. 0.                  ;
B = 0. 0.04 0.                  ;
N = 0. 0. 1.                    ;
C1 = CERC 100 B OC A            ;
C2 = C1 TOUR -90. OC N          ;
C3 = C1 ET C2                   ;
C4 = C3 TOUR -180. OC N         ;
C5 = C3 ET C4                   ;
OB = 0.01 0.038 0.                ;
CE1 = ELIM C5 0.0001            ;
PCEX = 0 0 0                    ;
PCEY = 0 0 0                    ;
PCEZ = 0 0 0                    ;
PBX = 0 0 0                     ;
PBY = 0 0 0                     ;
PCX = 0 0 0                     ;
PCY = 0 0 0                     ;
N1 = 0. 0. 1.                   ;
OEIL = 1. 1. 50.                ;
********************************
*      BASE MODALE              *
********************************
MAI1 = (CE1 ET OB)                        ;
CHPCE1 =
 MANU CHPO 6 (CE1 ET OC) 'UX' 1.0 'UY' 0.0 'UZ' 0.0
                         'RX' 0.0 'RY' 0.0 'RZ' 0.0   ;
CHPCE2 =
 MANU CHPO 6 (CE1 ET OC) 'UX' 0.0 'UY' 1.0 'UZ' 0.0
                         'RX' 0.0 'RY' 0.0 'RZ' 0.0   ;
CHPCE3 =
 MANU CHPO 6 (CE1 ET OC) 'UX' 0.0 'UY' 0.0 'UZ' 0.0
                         'RX' 0.0 'RY' 0.0 'RZ' 1.0   ;
CHPOB1  = MANU CHPO 6 OB 'UX' 1.0 'UY' 0.0 'UZ' 0.0
                         'RX' 0.0 'RY' 0.0 'RZ' 0.0   ;
CHPOB2  = MANU CHPO 6 OB 'UX' 0.0 'UY' 1.0 'UZ' 0.0
                         'RX' 0.0 'RY' 0.0 'RZ' 0.0   ;
CHPOC1  = MANU CHPO 6 OC 'UX' 1.0 'UY' 0.0 'UZ' 0.0
                         'RX' 0.0 'RY' 0.0 'RZ' 0.0   ;
CHPOC2  = MANU CHPO 6 OC 'UX' 0.0 'UY' 1.0 'UZ' 0.0
                         'RX' 0.0 'RY' 0.0 'RZ' 0.0   ;
* POINT LIGNE_FROTTEMENT
Z=0 0 0;
TBAS1 = TABLE 'BASE_MODALE'                           ;
TBAS1 . 'MODES' = TABLE 'BASE_DE_MODES'               ;
TBAS1 . 'MODES'. 'MAILLAGE' = CE1 ;
TBAS1 . 'MODES'. 1 = TABLE 'MODE'                     ;
TBAS1 . 'MODES'. 1 . 'POINT_REPERE'    =  PCEX        ;
TBAS1 . 'MODES'. 1 . 'FREQUENCE'              =  0.0;
TBAS1 . 'MODES'. 1 . 'MASSE_GENERALISEE'      =  40.;
TBAS1 . 'MODES'. 1 . 'DEFORMEE_MODALE'  = CHPCE1      ;
TBAS1 . 'MODES'. 1 .numero_mode = 1 ;
* TBAS1 . 'MODES'. 2 = TABLE 'MODE'                   ;
* TBAS1 . 'MODES'. 2 . 'POINT_REPERE'    =  PCEY      ;
* TBAS1 . 'MODES'. 2 . 'FREQUENCE'              =  0.0;
  TBAS1 . 'MODES'. 2 . 'MASSE_GENERALISEE'      =  40.;
  TBAS1 . 'MODES'. 2 . 'DEFORMEE_MODALE'  = CHPCE2      ;
  TBAS1 . 'MODES'. 2 .numero_mode = 2 ;

  TBAS2 = TABLE 'BASE_MODALE'                           ;
  TBAS2 . 'MODES' = TABLE 'BASE_DE_MODES'               ;
  TBAS2 . 'MODES'. 'MAILLAGE' = (OB et Z);              ;
  TBAS2 . 'MODES'. 1 = TABLE 'MODE'                     ;
  TBAS2 . 'MODES'. 1 . 'POINT_REPERE'    =  PBX         ;
  TBAS2 . 'MODES'. 1 . 'FREQUENCE'              =  0.0;
  TBAS2 . 'MODES'. 1 . 'MASSE_GENERALISEE'      =  40.;
  TBAS2 . 'MODES'. 1 . 'DEFORMEE_MODALE'  = CHPOB1      ;
  TBAS2 . 'MODES'. 1 .numero_mode = 1 ;
  TBAS2 . 'MODES'. 2 = TABLE 'MODE'                     ;
  TBAS2 . 'MODES'. 2 . 'POINT_REPERE'    =  PBY         ;
  TBAS2 . 'MODES'. 2 . 'FREQUENCE'              =  0.0;
  TBAS2 . 'MODES'. 2 . 'MASSE_GENERALISEE'      =  40.;
  TBAS2 . 'MODES'. 2 . 'DEFORMEE_MODALE'  = CHPOB2      ;
  TBAS2 . 'MODES'. 2 .numero_mode = 2 ;
  TBAST1 = TABLE 'ENSEMBLE_DE_BASES';
  TBAST1.1=TBAS1;
  TBAST1.2=TBAS2;
  * POINT CERCLE_FROTTEMENT
  MAI2 = OC ET OB                ;
  TBAS3 = TABLE 'BASE_MODALE'                             ;
  TBAS3 . 'MODES' = TABLE 'BASE_DE_MODES'                 ;
  TBAS3 . 'MODES'. 'MAILLAGE' = (OC et z);
  TBAS3 . 'MODES'. 1 = TABLE 'MODE'                       ;
  TBAS3 . 'MODES'. 1 . 'POINT_REPERE'    =  PCX           ;
  TBAS3 . 'MODES'. 1 . 'FREQUENCE'              =  0.0 ;
  TBAS3 . 'MODES'. 1 . 'MASSE_GENERALISEE'      =  40. ;
  TBAS3 . 'MODES'. 1 . 'DEFORMEE_MODALE'  = CHPOC1        ;
  TBAS3 . 'MODES'. 1 .numero_mode = 1 ;
  TBAS3 . 'MODES'. 2 = TABLE 'MODE'                       ;
  TBAS3 . 'MODES'. 2 . 'POINT_REPERE'    =  PCY           ;
  TBAS3 . 'MODES'. 2 . 'FREQUENCE'              =  0.0 ;
  TBAS3 . 'MODES'. 2 . 'MASSE_GENERALISEE'      =  40. ;
  TBAS3 . 'MODES'. 2 . 'DEFORMEE_MODALE'  = CHPOC2        ;
  TBAS3 . 'MODES'. 2 .numero_mode = 2 ;
  TBAST2 = TABLE 'ENSEMBLE_DE_BASES';
  TBAST2.1 = TBAS3;
  TBAST2.2 =TBAS2;
  *****************************************
  *               CHARGEMENT              *
  *****************************************
  *****************************************
  *               TEMPS                   *
  *****************************************
  PDT = 2.5E-5 ;
  NPDT = 400;
  NINS = 1;
  *********************************
  *      CONDITIONS INITIALES     *
  *********************************
  U0 = 5.                        ;
  V0 = 10.                       ;
  TINIT = TABLE 'INITIAL'        ;
  TINIT.'VITESSE'=
           (MANU CHPO PBX 1 alfa U0 'NATURE' 'DIFFUS') ET
           (MANU CHPO PBY 1 alfa V0 'NATURE' 'DIFFUS');

  *********************************
  *   LIAISON LIGNE_LIGNE         *
  *      FROTTEMENT               *
  *********************************
```

```
TLIA1 = TABLE 'LIAISON'         ;
TLB1 = TABLE 'LIAISON_B'                ;
TLIA1 .'LIAISON_B' = TLB1               ;
*
CHRA=MANU 'CHPO' 1 OB 'RAIN' 100000000.;
CHAM=MANU 'CHPO' 1 OB 'AMON' 63245.5;
*
TL1 = TABLE 'LIAISON_ELEMENTAIRE' ;
TL1.'TYPE_LIAISON'= MOT 'LIGNE_LIGNE_FROTTEMENT';
TL1.'LIGNE_MAITRE'              =  CE1          ;
TL1.'LIGNE_ESCLAVE'             =  OB           ;
TL1.'COEFFICIENT_GLISSEMENT'    = 20.0          ;
TL1.'COEFFICIENT_ADHERENCE'     = 20.0          ;
TL1.'RAIDEUR_TANGENTIELLE'      = -100. ;
TL1.'RAIDEURS'                  = CHRA;
TL1.'AMORTISSEMENT_TANGENTIEL'  = 0.01  ;
*TL1.'AMORTISSEMENTS'           =CHAM;
TL1.'NORMALE'                   = N1;
*TL1.'SYMETRIE'                  = 'GLOBALE';
TLB1.1 = TL1                    ;
*********************************
*   LIAISON POINT_CERCLE        *
*       FROTTEMENT              *
*********************************
TLIA2 = TABLE 'LIAISON'         ;
TLB2 = TABLE 'LIAISON_B'                ;
TLIA2 .'LIAISON_B' = TLB2               ;
*
TL2 = TABLE 'LIAISON_ELEMENTAIRE' ;
TL2.'TYPE_LIAISON'= MOT 'POINT_CERCLE_MOBILE';
TL2.'POINT'                     =  OB           ;
TL2.'CERCLE'                    =  OC           ;
TL2.'COEFFICIENT_GLISSEMENT'    = 20.0          ;
TL2.'COEFFICIENT_ADHERENCE'     = 20.0          ;
TL2.'RAIDEUR_TANGENTIELLE'      = -100. ;
TL2.'RAIDEUR'                   = 100000000.    ;
TL2.'AMORTISSEMENT_TANGENTIEL'  = 0.01  ;
*TL2.'AMORTISSEMENT'            =63245.5        ;
TL2.'NORMALE'                   = N1            ;
TL2.'RAYON'                     = 0.04          ;
TLB2.1 = TL2                    ;
*********************************
*       SORTIES                 *
*********************************
TSORT1 = TABLE 'SORTIE'         ;
TSORT2 = TABLE 'SORTIE'         ;
TSOR1 = TABLE 'SORTIE'          ;
TSORT1.'VARIABLE'       = TSOR1 ;
TSORT2.'VARIABLE'       = TSOR1 ;
TSOR1.'DEPLACEMENT'     = VRAI  ;
TSOR1.'VITESSE'         = FAUX  ;

TSOR2 = TABLE 'SORTIE'          ;
TSORT1.'LIAISON_B'      = TSOR2 ;
TSOR2.TL1               = FAUX  ;
TSOR3 = TABLE 'SORTIE'          ;
TSORT2.'LIAISON_B'      = TSOR3 ;
TSOR3.TL2               = FAUX  ;
*********************************
*   OPERATEUR DYNE              *
*********************************
TRESU11 = DYNE  'DIFFERENCES_CENTREES' TBAST1  TSORT1
          TLIA1 TINIT NPDT PDT NINS   ;
TRESU2 =  DYNE  'DIFFERENCES_CENTREES' TBAST2  TSORT2
          TLIA2 TINIT NPDT PDT NINS   ;

*
EV5 = EVOL rouge 'RECO' TRESU11 TBAST1 'DEPL' OB 'UX';
EV6 = EVOL rouge 'RECO' TRESU11 TBAST1 'DEPL' OB 'UY';
EV7 = EVOL 'RECO' TRESU2 TBAST2 'DEPL' OB 'UX';
EV8 = EVOL 'RECO' TRESU2 TBAST2 'DEPL' OB 'UY';

*dess (ev5 et ev7) ; dess (ev6 et ev8) ;

TX1 =EXTR EV5 'ORDO';
TX2 =EXTR EV7 'ORDO';
TY1 =EXTR EV6 'ORDO';
TY2 =EXTR EV8 'ORDO';
NFOIS= dime tresu11.temps_de_sortie;
max_x = 0.;
max_y = 0.;
*
*recherche des maximums
REPETER BLOC1 NFOIS;
x1 = EXTR TX1 (&BLOC1);
x2 = EXTR TX2 (&BLOC1);
y1 = EXTR TY1 (&BLOC1);
y2 = EXTR TY2 (&BLOC1);
ecart_x = ABS(x1 - x2);
ecart_y = ABS(y1 - y2);
SI (ecart_x > max_x);
max_x = ecart_x;
FINSI;
SI (ecart_y > max_y);
max_y = ecart_y;
FINSI;
FIN BLOC1;

* validation par rapport aux resultats numeriques
*              d'une certaine version

SI (max_x > 0.0018);
 ERREUR 5;
SINON;
 SI (max_y > 0.0012);
  ERREUR 5;
 SINON;
  ERREUR 0;
 FINSI;
FINSI;
FIN;
```

Test dy_devo8 **Comments**

1. **Linkage LIGNE_LIGNE_FROTTEMENT**

```
TLIA1 = TABLE 'LIAISON' ;
TLB1 = TABLE 'LIAISON_B' ;
TLIA1 .'LIAISON_B' = TLB1 ;
CHRA = MANU 'CHPO' 1 OB 'RAIN' 100000000.;
TL1 = TABLE 'LIAISON_ELEMENTAIRE' ;
TL1.'TYPE_LIAISON'= MOT 'LIGNE_LIGNE_FROTTEMENT';
TL1.'LIGNE_MAITRE' = CE1 ;
TL1.'LIGNE_ESCLAVE' = OB ;
TL1.'COEFFICIENT_GLISSEMENT' = 20.0 ;
TL1.'COEFFICIENT_ADHERENCE' = 20.0 ;
TL1.'RAIDEUR_TANGENTIELLE' = -100. ;
TL1.'RAIDEURS' = CHRA ;
TL1.'AMORTISSEMENT_TANGENTIEL' = 0.01 ;
TL1.'NORMALE' = N1 ;
TL1.'SYMETRIE' = 'GLOBALE';
TLB1.1 = TL1 ;
```

The LIGNE_LIGNE_FROTTEMENT linkage is used for impacts of a slave line (TL1.'LIGNE_ESCLAVE') against a master line (TL1.'LIGNE_MAITRE'). The slave line can be restricted to one point like here. The vector N1 (TL1.'NORMALE') represents the normal perpendicular to the contact plane. The computation tests the penetration of the slave nodes in the master surface (TL1.'SYMETRIE').
The linkage takes into consideration the friction, also specific data must be given : the sliding coefficient (TL1.'COEFFICIENT_GLISSEMENT'), the adherence stress coefficient (TL1.'COEFFICIENT_ADHERENCE'), the tangential stiffness (TL1.'RAIDEUR_TANGENTIELLE'), the tangential damping (TL1.'AMORTISSEMENT_TANGENTIEL'). The impact stiffness (TL1.'RAIDEURS') is given in CHPOINT form (if the slave line is not restricted to one point, it is necessary to define the impact stiffness for each slave node).

2. **Linkage POINT_CERCLE_MOBILE**

```
TLIA2 = TABLE 'LIAISON' ;
TLB2 = TABLE 'LIAISON_B' ;
TLIA2 .'LIAISON_B' = TLB2 ;
TL2 = TABLE 'LIAISON_ELEMENTAIRE' ;
TL2.'TYPE_LIAISON'= MOT 'POINT_CERCLE_MOBILE';
TL2.'POINT' = OB ;
TL2.'CERCLE' = OC ;
TL2.'COEFFICIENT_GLISSEMENT' = 20.0 ;
TL2.'COEFFICIENT_ADHERENCE' = 20.0 ;
TL2.'RAIDEUR_TANGENTIELLE' = -100. ;
TL2.'RAIDEUR' = 100000000. ;
TL2.'AMORTISSEMENT_TANGENTIEL' = 0.01 ;
TL2.'NORMALE' = N1 ;
TL2.'RAYON' = 0.04 ;
TLB2.1 = TL2 ;
```

The POINT_CERCLE_MOBILE linkage is used for impacts of a point (TL2.'POINT') against a mobile circle. The circle is simply described by its center (TL2.'CERCLE') and its radius (TL2.'RAYON'). The given normal (TL2.'NORMALE') is perpendicular to the circle plane.
The linkage takes into consideration the friction. Specific data are the same as the previous linkage except for the impact stiffness which is here a real (because it is related to one point).

3. **Loop counter**

```
REPETER BLOC1 NFOIS;
x1 = EXTR TX1 (&BLOC1);
x2 = EXTR TX2 (&BLOC1);
y1 = EXTR TY1 (&BLOC1);
y2 = EXTR TY2 (&BLOC1);
...
FIN BLOC1 ;
```

&BLOC1 is an integer which contains the number of times the loop BLOC1 has been in execution, it can therefore be used as a loop counter.

12.14 Test dyne01 Description sheet

Test name dyne01

Calculation type MECHANICS DYNAMICS 3D
POLYNOMIAL LINKAGE

Finite element type

Topic
Using a polynomial linkage and test of restart computation.

The polynomial linkage calculates a force applied to a mode i and composed of a set of contributions stemming from other modes.

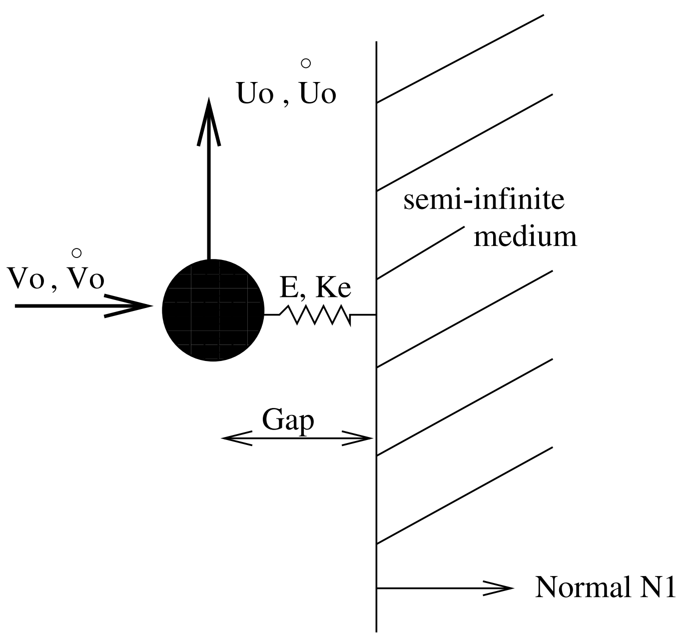
There is no geometry in this test, the computation is done in a modal basis, the used points are only modal points.

Goal Comparison of displacement, velocity and acceleration between both computations.

Reference

Version 97' customer version

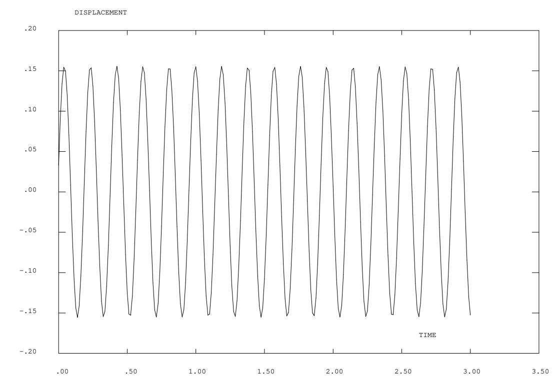
Model description



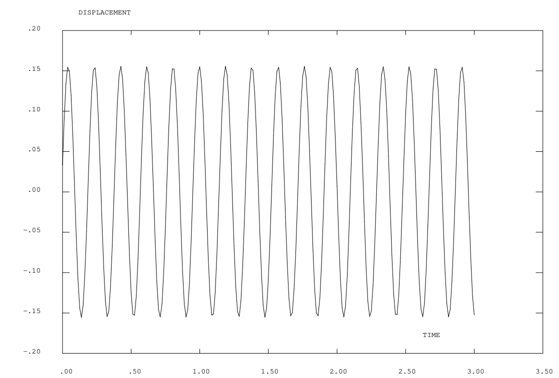
$$
\begin{aligned}
Gap &= 1 \\
K_e &= 0 \\
C &= 0 \\
U_o &= 0.033 \\
V_o &= 0.54 \\
\dot{U}_o &= 5 \\
\dot{V}_o &= -0.9
\end{aligned}
$$

Test dyne01 Results

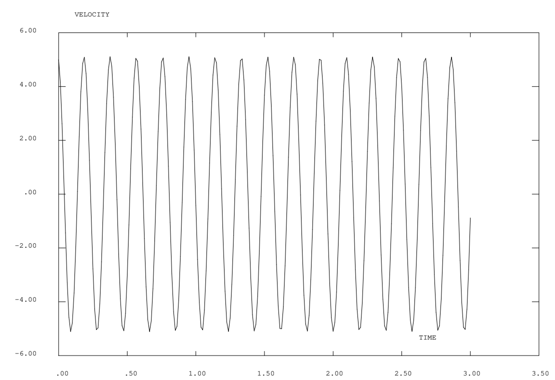
CASTEM figures



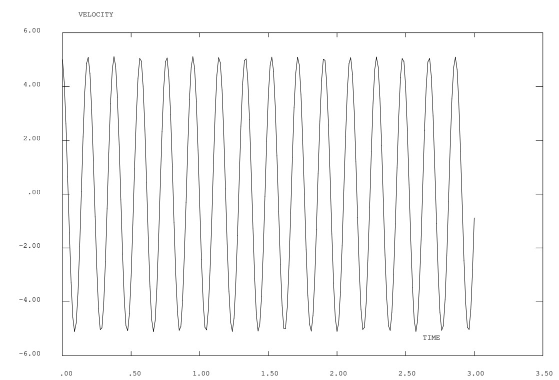
COMPUTATION 1 : DISPLACEMENT



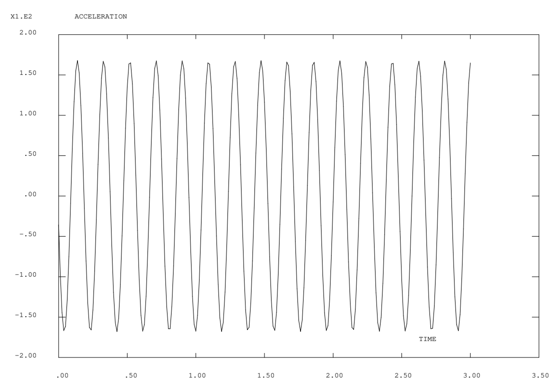
COMPUTATION 1 : DISPLACEMENT



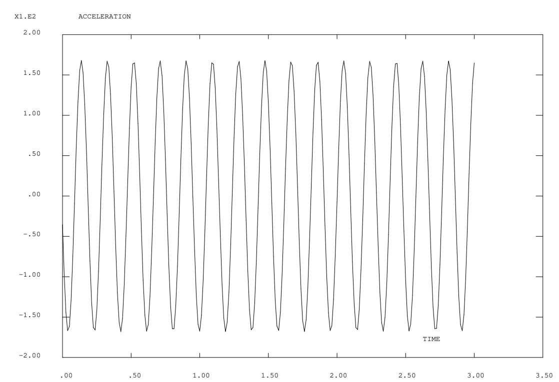
COMPUTATION 1 : VELOCITY



COMPUTATION 2 : VELOCITY



COMPUTATION 1 : ACCELERATION



COMPUTATION 2 : ACCELERATION

```
*          Test Dyne01.dgibi: Jeux de données       *
*          ---------------------------------        *
*                                                   *
*--------------------------------------------------
* Test dyne01.dgibi
*--------------------------------------------------
TITRE
' DYNE liaison POLYNOMIALE: test reprise de calcul';
OPTION DIME 3 MODE TRID ECHO 0 ;
*
* Creation des points modaux
*
P0 = 0. 0. 0. ;
PX = 0. 0. 0. ;
PY = 0. 0. 0. ;
*
* Données
*
MG = 1.25 ;
FR = 5.23 ;
T = 1. / FR ;
DEFMX = MANU 'CHPO' P0 3 UX 1. UY 0. UZ 0. ;
DEFMY = MANU 'CHPO' P0 3 UX 0. UY 1. UZ 0. ;
*
* Table de bases de modes
*
TAB1 = TABLE 'BASES_DE_MODE' ;
TABMO1 = TABLE 'MODE' ;
TABMO1.'POINT_REPERE' = PX ;
TABMO1.'FREQUENCE' = FR ;
TABMO1.'MASSE_GENERALISEE' = MG ;
TABMO1.'DEFORMEE_MODALE' = DEFMX ;
TAB1.1 = TABMO1 ;
*
TABMO2 = TABLE 'MODE' ;
TABMO2.'POINT_REPERE' = PY ;
TABMO2.'FREQUENCE' = FR ;
TABMO2.'MASSE_GENERALISEE' = MG ;
TABMO2.'DEFORMEE_MODALE' = DEFMY ;
TAB1.2 = TABMO2 ;
*
TAB1.'MAILLAGE' = P0 ;
TBAS = TABLE 'BASE_MODALE' ;
TBAS.'MODES' = TAB1 ;
*
* Table des conditions initiales
*
CPID = (MANU 'CHPO' PX 1 ALFA .033 ) +
       (MANU 'CHPO' PY 1 ALFA .54 ) ;
CPIV = (MANU 'CHPO' PX 1 ALFA 5.) +
       (MANU 'CHPO' PY 1 ALFA -0.9) ;
TINIT = TABLE 'INITIAL' ;
TINIT.'DEPLACEMENT' = CPID ;
TINIT.'VITESSE'    = CPIV ;
TINIT2 = TINIT ;
*
* Tables des liaisons
*
EPS1 = -1.E-9 ;
TLIAI = TABLE 'LIAISON' ;
TLA   = TABLE 'LIAISON_A' ;
*
TLB = TABLE 'LIAISON_B' ;
TLIAI.'LIAISON_B' = TLB ;
TLB1 = TABLE 'LIAISON_ELEMENTAIRE' ;
TLB1.'TYPE_LIAISON' = MOT 'POINT_PLAN' ;
TLB1.'SUPPORT' = PX ;
TLB1.'NORMALE' = (1.0 0.0 0.0) ;
TLB1.'RAIDEUR' = 0. ;
TLB1.'JEU' = 1.0 ;
TLB1.'AMORTISSEMENT' = 0. ;
TLB.1 = TLB1 ;
*
TLIAI.'LIAISON_A' = TLA ;
TL1   = TABLE 'LIAISON_ELEMENTAIRE' ;
*
TL1.'TYPE_LIAISON' = MOT 'POLYNOMIALE' ;
TL1.'SUPPORT'     = PX ;
TL1.'COEFFICIENT'  = .67 ;
TL2 = TABLE 'CONTRIBUTION' ;
TL1.PY = TL2 ;
TL2.'EXPOSANT_DEPLACEMENT' = 1.2 ;
TL2.'RETARD_DEPLACEMENT'   = 0.01 ;
TL2.'EXPOSANT_VITESSE'     = 0. ;
TL2.'RETARD_VITESSE'       = 0. ;
TL2.'JEU_DEPLACEMENT'      = 0. ;
TL2.'JEU_VITESSE'          = 0. ;
TLA.1 = TL1 ;
TL3   = TABLE 'LIAISON_ELEMENTAIRE' ;
TL3.'TYPE_LIAISON' = MOT 'POLYNOMIALE' ;
TL3.'SUPPORT'     = PX ;
TL3.'COEFFICIENT'  = .67 ;
TL4 = TABLE 'CONTRIBUTION' ;
TL3.PY = TL4 ;
TL4.'EXPOSANT_DEPLACEMENT' = 1.2 ;
TL4.'RETARD_DEPLACEMENT'   = 0. ;
TL4.'EXPOSANT_VITESSE'     = 0. ;
TL4.'RETARD_VITESSE'       = 0. ;
TL4.'JEU_DEPLACEMENT'      = EPS1 ;
TL4.'JEU_VITESSE'          = 0. ;
TLA.2 = TL3 ;
*
* Table de sortie des variables generalisees
*
TSORT = TABLE 'SORTIE' ;
TSORV = TABLE 'VARIABLE' ;
TSORV.'DEPLACEMENT'   = vrai ;
TSORV.'VITESSE'       = vrai ;
TSORV.'ACCELERATION'  = vrai ;
TSORT.'VARIABLE'      = TSORV ;
*
* Table de sortie variables de liaison sur base A
*
TSORL = TABLE 'LIAISON_A' ;
TSORL.TL1 = TABLE ;
TSORL.TL1.'FORCE_POLYNOMIALE' = vrai ;
TSORL.TL3 = TABLE ;
TSORL.TL3.'FORCE_POLYNOMIALE' = vrai ;
TSORT.'LIAISON_A' = TSORL ;
*
* Table de sortie variables de liaison sur base B
*
TSORB = TABLE 'LIAISON_B' ;
TSORB.TLB1 = TABLE ;
TSORB.TLB1.'FORCE_DE_CHOC' = VRAI ;
TSORT.'LIAISON_B' = TSORB ;
*
* Donnee des valeurs temporelles : NT & DT
*
NT = 1200 ;
DT = 0.0025 ;
TT = NT * DT ;
NPC = 5 ;
LST = PROG 0. PAS (DT * NPC) NPAS (NT / NPC) ;
*
*------> appel a DYNE avec NT pas de temps
```

```
*
TRES1 = DYNE 'DE_VOGELAERE'
           TBAS TLIAI TINIT NT DT NPC TSORT ;
*
*------> appel a DYNE avec NT/2 pas de temps
*         puis reprise avec NT/2 pas supplementaires
*
NT2 = ENTIER(NT/2) ;
TRES21 = DYNE 'DE_VOGELAERE'
         TBAS TLIAI TINIT NT2 DT NPC TSORT ;
*
TREP = TRES21.'REPRISE' ;
TINIT.'REPRISE' = TREP ;
*
TRES22 = DYNE 'DE_VOGELAERE'
          TBAS TLIAI TINIT NT2 DT NPC TSORT ;
*
LIS1  =  TRES1.'TEMPS_DE_SORTIE' ;
LIS21 = TRES21.'TEMPS_DE_SORTIE' ;
LIS22 = TRES22.'TEMPS_DE_SORTIE' ;
*
* Deplacement / vitesse / acceleration
*
I = 0 ;
NF1 = ENTIER (NT / NPC)  + 1 ;
REPETER BOUC1 NF1 ;
   I = I + 1 ;
   CHD1 = TRES1.I.'DEPLACEMENT' ;
   DEP1 = EXTR CHD1 ALFA PX ;
   CHV1 = TRES1.I.'VITESSE' ;
   VIT1 = EXTR CHV1 ALFA PX ;
   CHA1 = TRES1.I.'ACCELERATION' ;
   ACC1 = EXTR CHA1 ALFA PX ;
   SI (I EGA 1) ;
      LD1 = PROG DEP1 ;
      LV1 = PROG VIT1 ;
      LA1 = PROG ACC1 ;
   SINON ;
      LD1 = LD1 ET (PROG DEP1) ;
      LV1 = LV1 ET (PROG VIT1) ;
      LA1 = LA1 ET (PROG ACC1) ;
   FINSI ;
FIN BOUC1 ;
*
I = 0 ;
NF21 = ENTIER((NT / NPC) / 2) + 1 ;
REPETER BOUC21 NF21 ;
   I = I + 1 ;
   CHD21 = TRES21.I.'DEPLACEMENT' ;
   DEP21 = EXTR CHD21 ALFA PX ;
   CHV21 = TRES21.I.'VITESSE' ;
   VIT21 = EXTR CHV21 ALFA PX ;
   CHA21 = TRES21.I.'ACCELERATION' ;
   ACC21 = EXTR CHA21 ALFA PX ;
   SI (I EGA 1) ;
      LD21 = PROG DEP21 ;
      LV21 = PROG VIT21 ;
      LA21 = PROG ACC21 ;
   SINON ;
      LD21 = LD21 ET (PROG DEP21) ;
      LV21 = LV21 ET (PROG VIT21) ;
      LA21 = LA21 ET (PROG ACC21) ;
   FINSI ;
FIN BOUC21 ;
I = 0 ;
NF22 = ENTIER((NT / NPC) / 2) ;
REPETER BOUC22 NF22 ;
   I = I + 1 ;
   CHD22 = TRES22.I.'DEPLACEMENT' ;
   DEP22 = EXTR CHD22 ALFA PX ;
   CHV22 = TRES22.I.'VITESSE' ;
   VIT22 = EXTR CHV22 ALFA PX ;
   CHA22 = TRES22.I.'ACCELERATION' ;
   ACC22 = EXTR CHA22 ALFA PX ;
   SI (I EGA 1) ;
      LD22 = PROG DEP22 ;
      LV22 = PROG VIT22 ;
      LA22 = PROG ACC22 ;
   SINON ;
      LD22 = LD22 ET (PROG DEP22) ;
      LV22 = LV22 ET (PROG VIT22) ;
      LA22 = LA22 ET (PROG ACC22) ;
   FINSI ;
FIN BOUC22 ;
LD2 = LD21 ET LD22 ;
LV2 = LV21 ET LV22 ;
LA2 = LA21 et LA22 ;
*
* Comparaison des deplacements, vitesses et
* accelerations au dernier pas de calcul
*
LD2 = LD21 ET LD22 ;
DFIN1 = EXTRAIRE LD1 NF1 ;
DFIN2 = EXTRAIRE LD22 NF22 ;
ERD = ABS ( (DFIN1 - DFIN2) / DFIN1 ) ;
BOLD = (ERD <EG 0.001 ) ;
*
VFIN1 = EXTRAIRE LV1 NF1 ;
VFIN2 = EXTRAIRE LV22 NF22 ;
ERV = ABS ( (VFIN1 - VFIN2 ) / VFIN1 ) ;
BOLV = ( ERV <EG 0.001 ) ;
*
AFIN1 = EXTRAIRE LA1 NF1 ;
AFIN2 = EXTRAIRE LA22 NF22 ;
ERA = ABS ( (AFIN1 - AFIN2 ) / AFIN1 ) ;
BOLA = ( ERA <EG 0.001 ) ;
*
SI ( BOLA et BOLV et BOLD ) ;
   ERREUR 0 ;
SINON ;
   ERREUR 5 ;
FINSI ;
*
FIN ;
```

Test dyne01 **Comments**

1. **Linkage POLYNOMIALE**

```
TLIAI.'LIAISON_A' = TLA ;
TL1 = TABLE 'LIAISON_ELEMENTAIRE' ;
TL1.'TYPE_LIAISON' = MOT 'POLYNOMIALE' ;
TL1.'SUPPORT' = PX ;
TL1.'COEFFICIENT' = .67 ;
TL2 = TABLE 'CONTRIBUTION' ;
TL1.PY = TL2 ;
TL2.'EXPOSANT_DEPLACEMENT' = 1.2 ;
TL2.'RETARD_DEPLACEMENT' = 0.01 ;
TL2.'EXPOSANT_VITESSE' = 0. ;
TL2.'RETARD_VITESSE' = 0. ;
TL2.'JEU_DEPLACEMENT' = 0. ;
TL2.'JEU_VITESSE' = 0. ;
TLA.1 = TL1 ;
```

The polynomial linkage calculates a force f_i applied to a mode i and composed of a set of contributions stemming from other modes j :

$$f_i = a_i[Q_j^b(t-T_j)\dot{Q}_j^c(t-T_j)][Q_k^d(t-T_k)\dot{Q}_k^e(t-T_k)]$$

with $Q_j =< q_j - J_d >$
$\dot{Q}_j =< \dot{q}_j - J_v >$

a_i is the modal coefficient (TL1.'COEFFICIENT').
Each modal contribution is defined in a CONTRIBUTION subtype table (TL2) which contains :
- the displacement exponent b (TL2.'EXPOSANT_DEPLACEMENT')
- the displacement term of delay d (TL2.'RETARD_DEPLACEMENT')
- the velocity exponent c (TL2.'EXPOSANT_VITESSE')
- the velocity term of delay e (TL2.'RETARD_VITESSE')
- the gap connected with the displacement J_d (TL2.'JEU_DEPLACEMENT')
- the gap connected with the velocity J_v (TL2.'JEU_VITESSE')
TL2 is given in TL1.PY where PY is the modal origin point.

12.15 Test vibr2 Description sheet

Test name vibr2

Calculation type MECHANICS DYNAMICS 2D AXISYMMETRICAL

Finite element type QUA8

Topic
Uniform radial mode shapes of a thick sphere.

Eigen frequencies and shapes of a thick sphere are calculated. Only the uniform radial vibration shapes are studied.

Only a sector of the sphere is meshed and computation is done with axisymmetrical mode.

Boundary conditions corresponding to this meshing are obtained by using the operator RELA : nodes of similar radius have similar displacements.

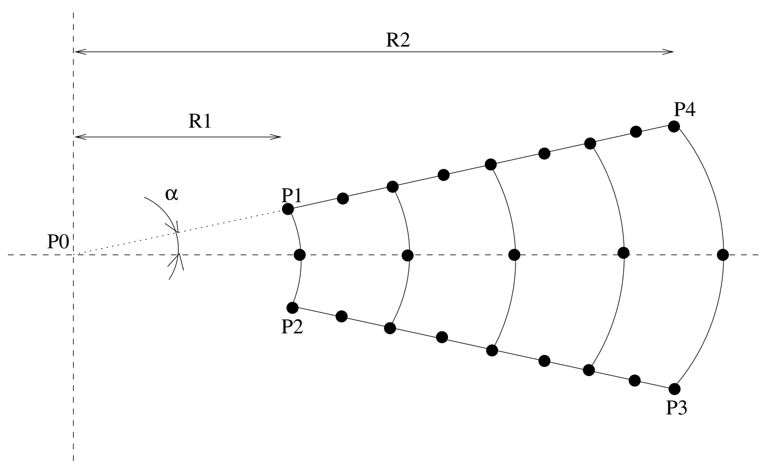
Goal Find the eigen frequencies of the sphere.

Reference Test NAFEMS number fv42.

Version 97' customer version

Model description

R_1	$= 1.8m$		ρ	$= 8000kg/m^3$
R_2	$= 6.m$		E	$= 2.10^{11}MPa$
α	$= 5^o$		ν	$= 0.3$

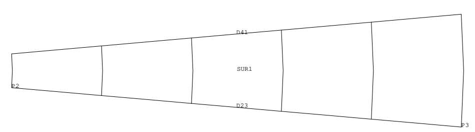


Test vibr2 **Results**

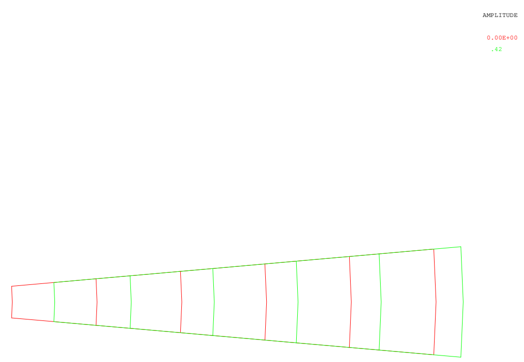
Results

	Theoretical freq (Hz)	CASTEM freq (Hz)	margin
Shape 1	369.91	368.62	0.35 %
Shape 2	838.03	834.92	0.37 %
Shape 3	1451.2	1447.5	0.25 %
Shape 4	2117.0	2123.6	0.31 %
Shape 5	2795.8	2841.8	1.64 %

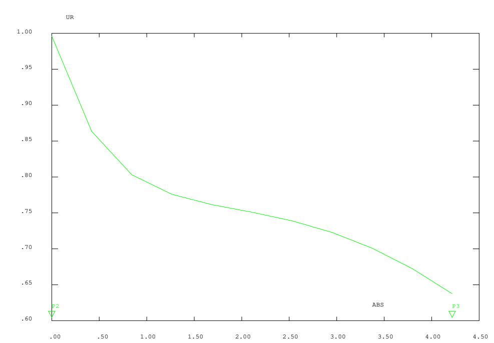
CASTEM figures



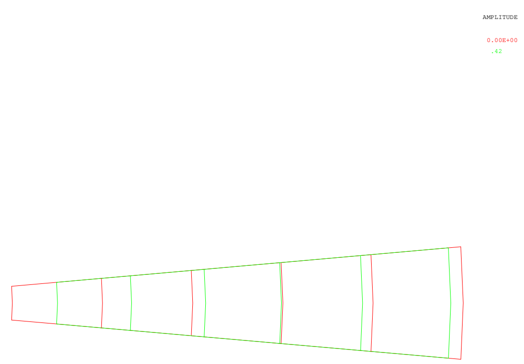
VIBR2 : MESHING



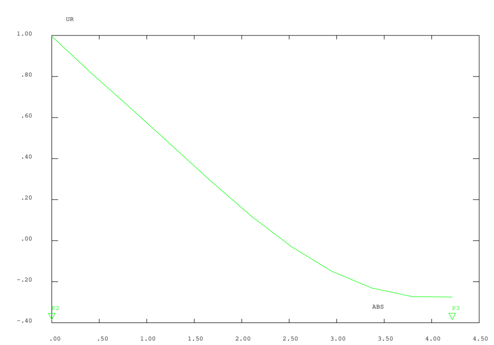
MODE NUMERO 1 FREQUENCE 368.617 HZ



Eigen shape : Radial displacement



MODE NUMERO 2 FREQUENCE 834.919 HZ



Eigen shape : Radial displacement

```
*          Test Vibr2.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
* CAS TEST DU 91/06/13     PROVENANCE : TEST
SAUT PAGE;
*
*****************************************************
*                                                   *
*  TEST VIBR2                                       *
*                                                   *
*  SPHERE EPAISSE : VIBRATION RADIALE UNIFORME      *
*                                                   *
*  Test NAFEMS , test numero fv42                   *
*                                                   *
*     On se propose de calculer les frequences et   *
*  modes propres d'une sphere epaisse. On ne s'inte- *
*  resse qu'aux modes de vibration radiale uniforme. *
*                                                   *
*     On ne maille qu'un secteur de la sphere et l'on *
*  travaille en mode axisymetrique.                 *
*                                                   *
*     Pour obtenir les conditions aux limites corres- *
*  pondantes, on utilise l'operateur RELA (les noeuds *
*  ayant le meme rayon ont le meme deplacement radial)*
*                                                   *
*****************************************************
*
OPTI ECHO 1;
*
OPTI DIME 2;
OPTI MODE AXIS;
OPTI ELEM QUA8;
*
GRAPH = 'N';
*---------------------------------------------------*
*                geometrie - maillage               *
*---------------------------------------------------*
*
ALPHA = 5.; COSALPHA = COS ALPHA; SINALPHA = SIN ALPHA;
*
P0 = 0. 0.;
P1 = 1.8 ( 1.8 * SINALPHA);
P2 = 1.8 (-1.8 * SINALPHA);
P3 = 6.0 (-6.0 * SINALPHA);
P4 = 6.0 ( 6.0 * SINALPHA);
*
D23 = P2 DROI 5 P3;
*
SUR1 = D23 ROTA 1 (2 * ALPHA) P0;
D41 = COTE 3 SUR1;
*
ELIM (SUR1 ET D23 ET D41) 0.001;
*
SI (NEG GRAPH 'N');
  TITR 'VIBR2 : MAILLAGE';
  TRAC QUAL SUR1;
FINSI;
*
*---------------------------------------------------*
*    modl - materiau - rigidite - masse             *
*---------------------------------------------------*
*
MOD1 = MODL SUR1 MECANIQUE QUA8;

MAT1 = MATR MOD1 YOUN 2.E11 NU 0.3 RHO 8E3;
RIG1 = RIGI MOD1 MAT1;
MAS1 = MASS MOD1 MAT1;
*
*---------------------------------------------------*
*                conditions aux limites             *
*---------------------------------------------------*
DIR1 = (-1 * SINALPHA) (-1 * COSALPHA);
DIR2 = (-1 * SINALPHA) COSALPHA;
*
CL1 = BLOQ DEPL DIRECTION DIR1 D23;
CL2 = BLOQ DEPL DIRECTION DIR2 D41;
CLI1 = CL1 ET CL2;
*
DIR3 = COSALPHA (-1 * SINALPHA);
DIR4 = COSALPHA SINALPHA;
*
LIST1 = LECT 27 22 17 23 18 24 19 25 20 26 21;
LIST2 = LECT  1  6  2  7  3  8  4  9  5 10 11;
*
I = 0;
REPETER BLOC1 11;
  I = I + 1;
  J = EXTR LIST1 I;
  K = EXTR LIST2 I;
  CL3 = RELA 1 DEPL DIRECTION DIR4 (NOEUD J)
         -   1 DEPL DIRECTION DIR3 (NOEUD K);
  CLI1 = CLI1 ET CL3;
FIN BLOC1;
*
LIST1 = LECT 27 17 18 19 20 21;
LIST2 = LECT 28 12 13 14 15 16;
*
I = 0;
REPETER BLOC2 6;
  I = I + 1;
  J = EXTR LIST1 I;
  K = EXTR LIST2 I;
  CL4 = RELA 1 DEPL DIRECTION DIR4 (NOEUD J)
         -   1 UR (NOEUD K);
  CLI1 = CLI1 ET CL4;
FIN BLOC2;
*
RIG1 = RIG1 ET CLI1;
*
*---------------------------------------------------*
*           calcul des frequences propres           *
*---------------------------------------------------*
*
* Utilisation de l'operateur VIBR.
*
FRE1 = TABLE;
FRE1.1 = 369.91;
FRE1.2 = 838.03;
FRE1.3 = 1451.2;
FRE1.4 = 2117.0;
FRE1.5  = 2795.8;
*
LIST1 = PROG FRE1.1 FRE1.2 FRE1.3 FRE1.4 FRE1.5;
*
RESUL = VIBR PROC LIST1 RIG1 MAS1;
*
*---------------------------------------------------*
*        extraction des resultats
*---------------------------------------------------*
*
MESS ' RESULTATS FV42.DGIBI';
MESS ' --------------------';
SAUT 2 LIGN;
MESS ' SPHERE EN VIBRATION RADIALE';
SAUT 2 LIGN;
*
```

```
FRE2 = TABLE; ERG = TABLE;
DEP = TABLE;  DEF = TABLE; EVO = TABLE;
*
I = 0;
REPETER BLOC3 5;
*
  I = I + 1;
  FRE2.I = TIRE RESUL FREQ RANG I;
  ERG.I = 100 * (ABS ((FRE1.I - FRE2.I) / FRE1.I));
  MESS ' MODE ' I;
  MESS ' ----------';
  MESS ' Frequence propre theorique :' FRE1.I 'Hz';
  MESS ' Frequence propre calculee  :' FRE2.I 'Hz';
  MESS '     Soit un ecart de : ' ERG.I '%';
  SAUT 1 LIGN;
*
* Trace facultatif des modes propres
*
  SI (NEG GRAPH 'N');
     DEP.I = TIRE RESUL DEPL RANG I;
     DEF0 = DEFO SUR1 DEP.I 0. ROUGE;
     DEF.I = DEFO SUR1 DEP.I VERT;
     TITR ' Mode propre : Deplacement radial';
     TRAC (DEF0 ET DEF.I);
     EVO.I = EVOL VERT CHPO DEP.I UR D23;
     DESSIN EVO.I;
  FINSI;
*
FIN BLOC3;
*
*------------------------------------------------------*
*                 code fonctionnement                  *
*------------------------------------------------------*
*
ERGMAX = MAXI (PROG ERG.1 ERG.2 ERG.3 ERG.4 ERG.5);
*
SAUT 2 LIGN;
SI (ERGMAX <EG 5.);
   ERRE 0;
SINON;
   ERRE 5;
FINSI;
*
SAUT 2 LIGN;
TEMPS;
SAUT 2 LIGN;
*
FIN;
*
```

Test vibr2 **Comments**

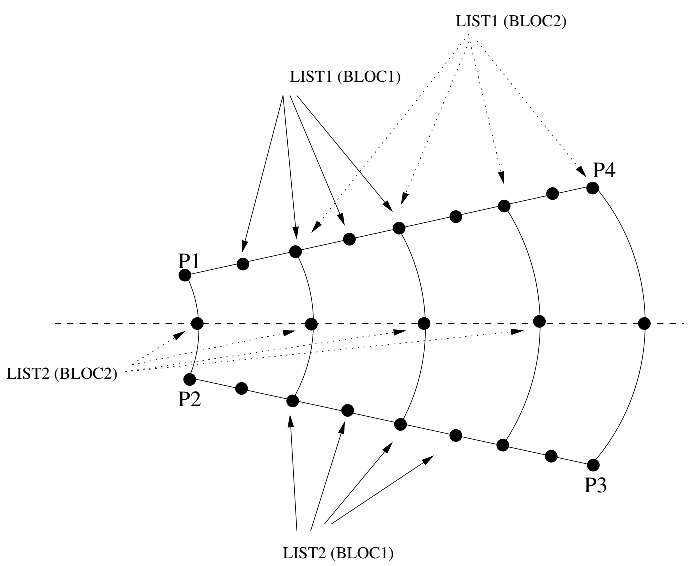
Boundary conditions

```
DIR1 = (-1 * SINALPHA) (-1 * COSALPHA);
DIR2 = (-1 * SINALPHA) COSALPHA;

CL1 = BLOQ DEPL DIRECTION DIR1 D23;
CL2 = BLOQ DEPL DIRECTION DIR2 D41;
CLI1 = CL1 ET CL2;
```

Only the radial vibrations are studied, so the normal displacements are constrained : DIR1 is the normal to D23 and DIR2 to D41.

Nodes of similar radius have similar radial displacements : this condition is given in loops BLOC1 and BLOC2. The different lists of integers correspond to the numbers of nodes (display the number of nodes when drawing the meshing). The loop BLOC1 is relative to nodes on D41 and D23. The loop BLOC2 is relative to nodes on D41 and middle nodes.



Chapter 13

NON LINEAR DYNAMICS

13.1 Test dyna_nl1 Description sheet

Test name dyna_nl1

Calculation type MECHANICS 2D
NON LINEAR GEOMETRICAL DYNAMICS

Finite element type BARR

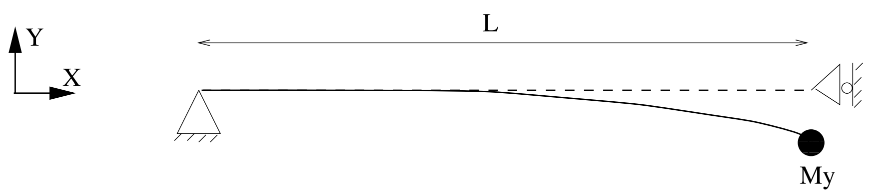
Topic
Free oscillations of an oscillator of Duffing type.

The bar is clamped at one end and prestressed. On the free end is imposed a vertical displacement. Then the bar is released and has free oscillations.

Goal Find the amplitude and period of the free oscillations of the bar.

Version 97' customer version

Model description



L	$= 1m$	E	$= 10^7 Pa$
δ	$= 0.2m$	ν	$= 0.3$
A	$= 1m^2$	ρ	$= 0kg/m^3$
M_y	$= 250kg$	F_{pre}	$= 500N$

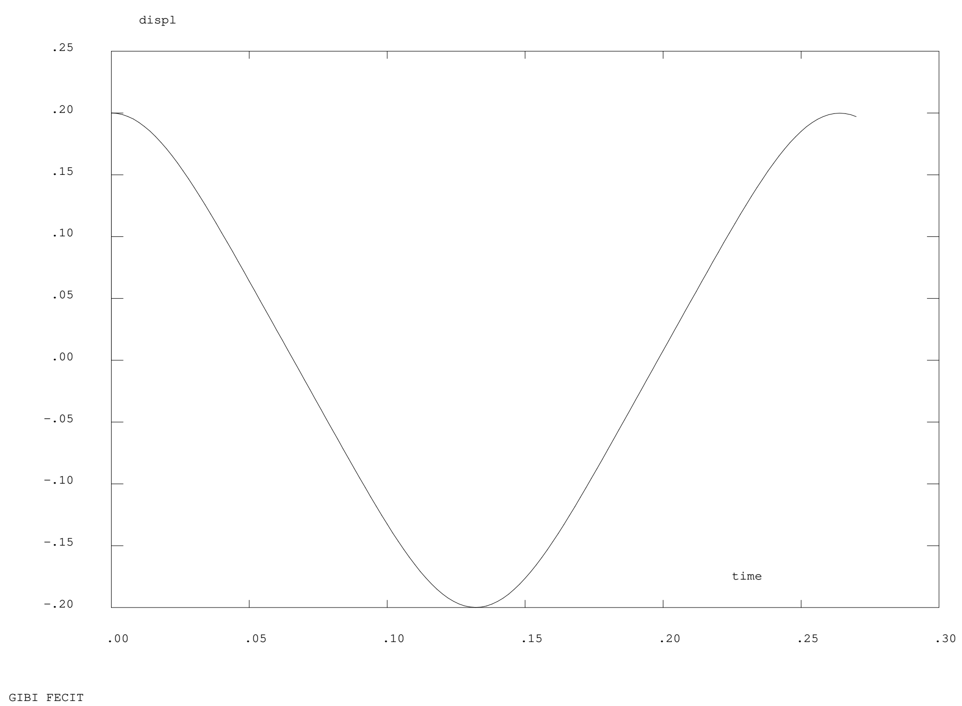
Test dyna_nl1

Results

Results

Margin in amplitude : 0.007%
Margin in period :0.3%

CASTEM figures



```
*        Test Dyna_nl1.dgibi: Jeux de données
*        -----------------------------------
*
complete = vrai;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
*************************************************
*******************************************************
*
*  dynamique non lineaire geometrique
*
*  oscillations libres d'un oscillateur de type Duffing
*
*  reference :J. ARGYRIS H.P. MLEJNEK
*            DYNAMICS OF STRUCTURES (p. 494)
*
*******************************************************
opti echo 0 ;
graph = faux ;


OPTI DIME 2 ELEM SEG2 MODE PLAN CONT ; ;
P1 = 0.    0.    ; P2 = 1. 0.  ;
ST = P1 D 1  P2 ;

MO = MODE ST  MECANIQUE  BARR     ;
MA1 = MATE MO YOUN 1E7  NU 0.3 RHO 0.;
CAR1 = CARA MO SECT 1. ;
MACA= MA1 ET CAR1;
MASS1 = MASS UY 250. P2 ;

CL1 = BLOQ UX uy  P1 ;
CL2 = BLOQ UX P2 ;
CL3 = BLOQ UY P2 ;
CL = CL1 ET CL2 ;


* precontrainte de 500 N

RI1 = RIGI MACA  MO ;
f1 = force fx 500 p2 ;
dep1 = reso (ri1 et cl1 et cl3) f1 ;
sig1 = sigm dep1 mo maca ;
*sig1 = manu 'CHML' mo  'EFFX' 500. 'TYPE' 'CONTRAIN'
*        'STRESSES' ;

* deplacement impose en statique

t1 = 1e-10 ;
FP11 = DEPI CL3 .2;
LIX1 = PROG 0. t1  100  ;
LIY1 = PROG 0. 1  1  ;
EV1 = EVOL MANU T LIX1 F(T) LIY1 ;
CHA1 = CHAR MECA FP11   EV1 ;
tab2 = table ;
tab2.grandes_rotations = vrai ;
TAB2.'BLOCAGES_MECANIQUES' = CL et cl3  ;
TAB2.'MODELE' = MO;
Tab2.'K_SIGMA' = VRAI ;
TAB2.'CARACTERISTIQUES' = MACA;
TAB2.'CHARGEMENT' = CHA1;
TAB2.'CONTRAINTES' = TABLE;
TAB2.'CONTRAINTES' . 0 =  SIG1 ;
LIS11 = PROG 0.  (.33*t1)  (.66*t1)  t1  ;
TAB2.'TEMPS_CALCULES' = LIS11;
PASAPAS TAB2 ;
* lacher et oscillations libres en dynamique
si complet;
t2 = .27 ;
sinon;
t2 = .02;
finsi;
dt = .002 ;
f1 = force fy 0. p2 ;
cha2 = char meca f1 ev1 ;
tab2.dynamique = vrai ;
tab2.masse_constante = mass1 ;
tab2.ftol = 1e-4 ;
TAB2.'BLOCAGES_MECANIQUES' = CL ;
LIS22 = PROG (t1 + dt)  pas dt  (t1 + t2)  ;
TAB2.'TEMPS_CALCULES' = LIS22;
TAB2.'CHARGEMENT' = CHA2;
PASAPAS TAB2 ;


NDIM = (DIME ( TAB2 . DEPLACEMENTS )) ;
PGY = PROG ;
REPETER TBOU2 NDIM;
 LEDEP = TAB2 . DEPLACEMENTS.  (&TBOU2 - 1);
 V   = EXTR LEDEP UY P2 ;
 PGY = PGY ET ( PROG V ) ;
FIN TBOU2 ;

ev1 = evol manu 'temps' (lis11 et lis22) 'depl' pgy ;
 si graph ;
   dess ev1 ;
 finsi ;

*recherche de l'amplitude et de la periode
si complet;
dim1 = enti ((t2 - .25)/dt) ;
maxuy = 0 ;
tt = .25 ;
 repeter bou1 dim1 ;
  tt = tt + dt ;
  uy1 = ipol tt (lis11 et lis22) pgy ;
    si (uy1 >  maxuy) ;
      maxuy = uy1 ;
      tp = tt ;
    finsi ;
 fin bou1 ;
valres = .2;
sinon;
tp =.2648;
valres =   .171 ;maxuy = pgy extr ( dime pgy);
finsi;


** periode theorique .264791269s
** amplitude theorique .2 ;
err1 = abs ((maxuy - valres)/valres * 100) ;
err2 = abs ((tp - .2648)/.2648 *100) ;
mess 'erreur en amplitude'err1'%';
mess 'erreur en periode'err2'%' ;
 si ((err1 > .1) ou (err2 > .5));
  erre 5 ;
 sinon ;
  erre 0 ;
 finsi ;
fin ;
```

Test dyna_nl1 **Comments**

1. **First step : Computation of prestress**

```
RI1 = RIGI MACA MO ;
f1 = force fx 500 p2 ;
dep1 = reso (ri1 et cl1 et cl3) f1 ;
sig1 = sigm dep1 mo maca ;
```

A horizontal force of 500 N is applied on the free end of the bar. To get a prestress we compute the stresses related to this load. Those results will be added in the following step.

2. **Second step : Imposed displacement**

```
FP11 = DEPI CL3 .2 ;
LIX1 = PROG 0. t1 100 ;
LIY1 = PROG 0. 1 1 ;
EV1 = EVOL MANU T LIX1 F(T) LIY1 ;
CHA1 = CHAR MECA FP11 EV1 ;
```

The operator DEPI imposes a specific value for the degree of freedom described in CL3. It means that UY-displacement of point P2 (related to CL3) will be equal to 0.2 m. The object created by DEPI is of CHPOINT type and must be added to the second member of the equation. To perform a dynamic computation the second member must be a function of time : the operator CHAR is then used to multiply the second member by a factor which is defined as a function of time (EV1).

3. **Second step : Computation by using PASAPAS subroutine**

```
tab2 = table ;
tab2.grandes_rotations = vrai ;
TAB2.'BLOCAGES_MECANIQUES' = CL et cl3 ;
TAB2.'MODELE' = MO;
Tab2.'K_SIGMA' = VRAI ;
TAB2.'CARACTERISTIQUES' = MACA;
TAB2.'CHARGEMENT' = CHA1;
TAB2.'CONTRAINTES' = TABLE;
TAB2.'CONTRAINTES' . 0 = SIG1 ;
LIS11 = PROG 0. (.33*t1) (.66*t1) t1 ;
TAB2.'TEMPS_CALCULES' = LIS11;
PASAPAS TAB2 ;
```

The non linear problem is solved by using PASAPAS subroutine. The single object necessary for the computation is a table (TAB2). The user must create this table which has indexes defined by keywords :
- TAB2.grandes_rotations : boolean equal to VRAI if the second-order terms must be taken into account for the strain tensor (in the case of large rotations or large deformations)
- TAB2.BLOCAGES_MECANIQUES : contains the boundary conditions
- TAB2.MODELE : contains the model object
- TAB2.K_SIGMA : boolean equal to VRAI if the matrix K_SIGMA is to be added to the iteration operator
- TAB2.CARACTERISTIQUES : contains material and geometrical caracteristics
- TAB2.CHARGEMENT : contains the load object (defined as a function of time)
- TAB2.CONTRAINTES : is a table : TAB2.'CONTRAINTES'.0 contains the stresses at the beginning of first step. The given values are the previous computed prestresses.
- TAB2.TEMPS_CALCULES : the given list of reals defines the values of time for which results are requested.

The subroutine is called by the following instruction : PASAPAS TAB2 .

4. **Third step : Release and free oscillations**

```
t2 = .27 ;
dt = .002 ;
f1 = force fy 0. p2 ;
cha2 = char meca f1 ev1 ;
tab2.dynamique = vrai ;
tab2.masse_constante = mass1 ;
tab2.ftol = 1e-4 ;
TAB2.'BLOCAGES_MECANIQUES' = CL ;
LIS22 = PROG (t1 + dt) pas dt (t1 + t2) ;
TAB2.'TEMPS_CALCULES' = LIS22;
TAB2.'CHARGEMENT' = CHA2;
PASAPAS TAB2 ;
```

The free oscillations are simulated by a zero force.
TAB2.dynamique is a boolean. If its value is true, it means that it is a dynamic analysis.
TAB2.masse_constante defines a constant lumped mass matrix (mass1).
TAB2.ftol indicates that the effort equilibrium will be checked up to about 1e-4.

5. **Results extraction**

```
NDIM = (DIME ( TAB2 . DEPLACEMENTS )) ;
PGY = PROG ;
REPETER TBOU2 NDIM;
LEDEP = TAB2 . DEPLACEMENTS. (&TBOU2 - 1);
V = EXTR LEDEP UY P2 ;
PGY = PGY ET ( PROG V ) ;
FIN TBOU2 ;
ev1 = evol manu 'time' (lis11 et lis22) 'displ' pgy ;
dess ev1 ;
```

The results of PASAPAS subroutine are stored in TAB2 table : TAB2.DEPLACEMENTS is a table which has as many indexes as the number of requested saving times.
TAB2.DEPLACEMENTS.i contains the displacements computed for the ith step.

13.2 Test dyna_nl3 Description sheet

Test name dyna_nl1

Calculation type MECHANICS 3D
NON LINEAR DYNAMICS

Finite element type POUT, COQ3

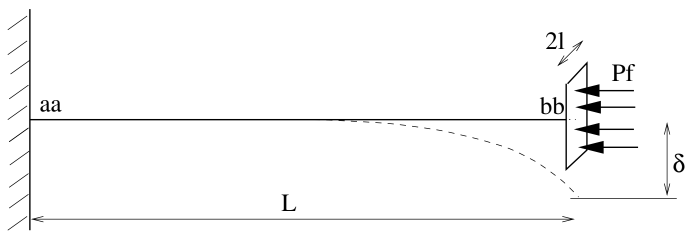
Topic
Follower pressure and wavering of a beam.

The response of a released beam is studied with an axial compression force which is a little higher than the theoretical value of wavering force ($Fcr = 20.05 * E * I/l^2$).
The beam is clamped-free. The follower force is described by an equivalent pressure which is applied on a small shell located on the end of the beam.
The system is disturbed by a small transversal displacement which is imposed at the free end.

Goal Find the maximum amplitude of the displacement of the free end.

Version 97' customer version

Model description



$L = 12$
$l = 0.01$
$\delta = 0.01$
$I_y = 2.8125.10^{-4}$
$I_z = 2.8125.10^{-4}$
$I_o = 10^{-5}$

$A_{beam} = 0.15$
$Th_{shell} = 0.01$
$E_{beam} = 2.1.10^7$
$\rho_{beam} = 7.8.10^{-3}$
$E_{shell} = 2.1.10^{10}$
$\rho_{shell} = 7.8.10^{-10}$
$\nu = 0.3$

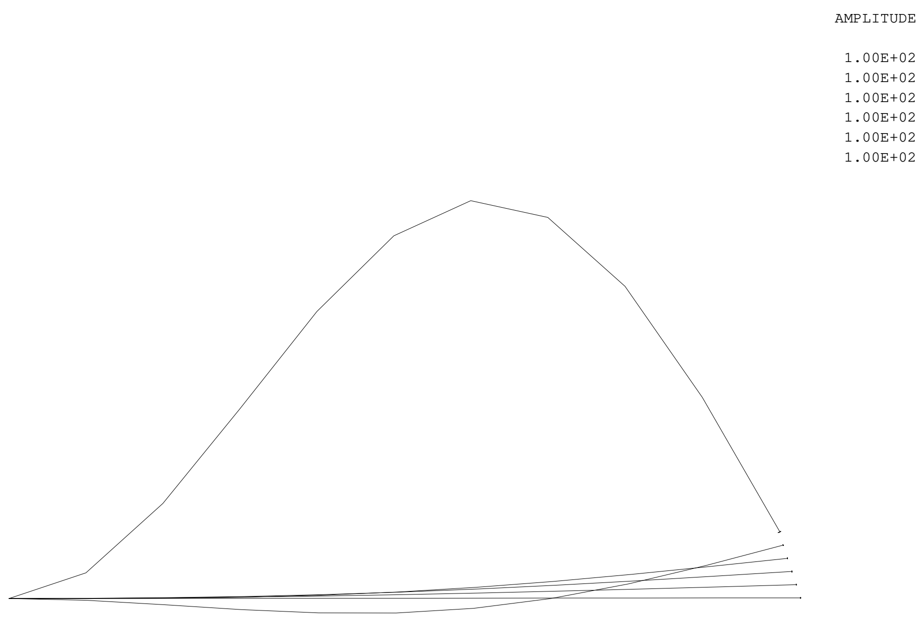
$P_f = 1.05 * [20.05 * E_{beam} * I_y/L^2]/(4l^2)$

Test dyna_nl3 **Results**

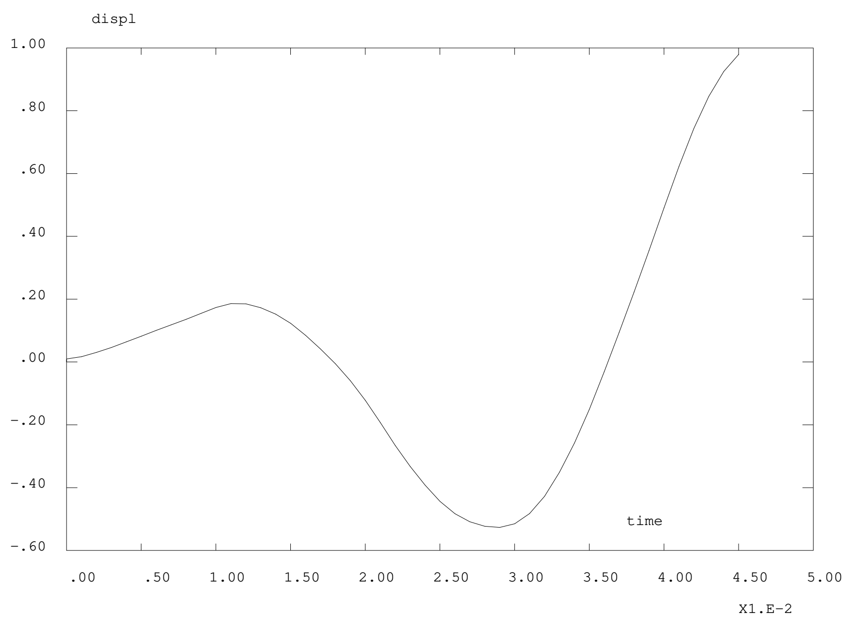
Results

Maximum amplitude (point bb) : 0.97959
Margin : 0.04%

CASTEM figures



Static computation



Dynamic computation

```
*         Test Dyna_nl3.dgibi: Jeux de données          *
*         ------------------------------------          *
*                                                       *
complet = vrai;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
***************************************************
*=======================================================
*       TEST DES LA PRESSION SUIVEUSE EN DYNAMIQUE
*
*       FLOTTEMENT D'UNE POUTRE ENCASTREE-LIBRE
*
* On etudie la reponse de la poutre à un lacher quand
* elle est soumise à une force de compression axiale
* legerement superieure à la valeur theorique de la
* force critique de flottement (Fcr = 20.05*E*I/l**2)
* On doit obtenir une reponse d'amplitude croissante.
*
*********************************************************
*                                                    ==
*    /|aa___________________bb                       ==
*    /|_______________________|<----- F              ==
*    /|                         cc                   ==
*           EQUIVALENT A     /|                      ==
*                        dd / |                      ==
*    /|aa_________________bb|<--                     ==
*    /|_____________________|<--  PRESSION           ==
*    /|                     |<--                     ==
*                        ee |/ ff                    ==
*                                                    ==
*                                                    ==
*=======================================================
graph = faux;
option dime 3 elem seg2 tri3 ;
*=======================================================
* definition de la geometrie
* la force suiveuse est modelisee par une pression équi
* valente appliquée sur une petite plaque fixée à l'ex
* trémité de la poutre
*=======================================================
aa = 0 0 0 ;
bb = 12 0 0;
cc = 12 0.01 0.01 ;
dd = 12 0.01 -0.01 ;
ee = 12 -0.01 -0.01 ;
ff = 12 -0.01 0.01 ;
dens1 = 10 ;
lig1 = aa d dens1 bb ;
lig2 = cc d 2 dd d 2 ee d 2 ff d 2 cc ;
surf1 = surf plan lig2 ;
geo = lig1 et surf1 ;
elim 1.e-4 geo ;
*======================================================
*definition des conditions aux limites
* encastrement à la base de la poutre et mouvement
* dans le plan xOy
*======================================================
cl1 = bloq rota depla aa ;
cl2 = bloq uz rx ry surf1 ;
cl3 = bloq bb uy ;
*=======================================================
*      definition du modele, du materiau et
*               des caracteristiques
*=======================================================
mo1 = mode lig1 mecanique pout ;
mo2 = mode surf1 mecanique coq3 ;
ma1 = mate mo1 youn 2.1e7 nu 0.3 rho 7.8e-3 ;
```

```
ma2 = mate mo2 youn 2.1e10 nu 0.3 rho 7.8e-10 ;
ca1 = cara mo1 sect 0.15 inry 2.8125e-4 inrz 2.8125e-4
                                               tors 1.e-5;
ca2 = cara mo2 epai 0.01 ;
mat1 = ma1 et ca1 ;
mat2 = ma2 et ca2 ;
mattotal = mat1 et mat2 ;
mototal = mo1 et mo2 ;
*=======================================================
* calcul des matrices de rigidite et de masse
*=======================================================
ri1 = rigi mototal mattotal ;
ritotal = ri1 et cl1 et cl2 ;
masst = masse mototal mattotal ;
*=======================================================
* chargement par pression equivalente a la force
* critique de flottement pondérée par un coefficient
* de 1.05 pour se placer legerement au dessus de
* l'instabilité.
*
* on impose un petit deplacement transverse pour
* perturber le systeme.
*=======================================================
fo2 = depi cl3 .01 ;
fo0 = manu chpo geo 1 p (2050781.25*1.05)
      nature discret ;
*=======================================================
*               calcul non lineaire
* calcul statique sous la force de compression et la
* force transversale de perturbation
*=======================================================
dt = 0.00001 ;
li1 = prog  0. dt ;
li2 = prog  0. 1 ;
ev = evol manu t (li1 ) f(t) (li2 )  ;
fcons = char dimp  fo2 ev ;
fncons = char psui fo0 ev ;
fco = fncons et fcons ;
lis1a = prog 0. pas (dt/5) dt ;


tab3 = table ;
tab3.grands_deplacements = vrai ;
tab3.chargement = fco ;
tab3.blocages_mecaniques = cl1  et cl2 et cl3 ;
tab3.modele =  mototal ;
tab3.caracteristiques = mattotal ;
tab3.temps_calcules =  lis1a ;
tab3.precision      =  1e-8  ;
tab3.deltaiter = 20 ;

pasapas tab3 ;

oeil = 0 0 10000 ;
dim1 = dime (tab3.deplacements) ;
lisx = prog  ; lisy = prog ;
 repeter bou1 dim1 ;
   n = &bou1 - 1 ;
   uy1 = extr tab3.deplacements.n 'UY' bb ;
   lisy = lisy et (prog uy1) ;
   lisx = lisx et (prog tab3.temps. n);
   def1 = defo tab3.deplacements.n  geo 100 ;
     si (&bou1 ega 1) ; deft = def1 ; sinon ;
     deft = deft et def1 ; finsi ;
 fin bou1 ;
ev1 = evol manu 'temps' lisx 'depl' lisy ;
si graph ; trac deft oeil cach anime ; finsi ;
```

```
*=======================================================
*           calcul non lineaire dynamique
*=======================================================
* definition de l'évolution du chargement
* durée de la phase de chargement statique :dt

ev2 = evol manu (prog dt 100) (prog 1 1 ) ;
*
fncons = char psui fo0 ev2 ;
si complet;
lis1c = prog dt pas .001 .045 ;
sinon;
lis1c = prog dt  pas .001 0.005;
finsi;
tab3.chargement = fncons ;
tab3.blocages_mecaniques = cl1  et cl2 ;
tab3.dynamique = vrai ;
tab3.temps_calcules = lis1c ;

pasapas tab3 ;

*=====================================================
* exploitation des resultats : deplacement de l'
* extrémité de la poutre (point bb)
*=====================================================

dim1 = dime (tab3.deplacements) ;
lisx = prog  ; lisy = prog ;
 repeter bou1 dim1 ;
   n = &bou1 - 1 ;
   uy1 = extr tab3.deplacements.n 'UY' bb ;
   lisy = lisy et (prog uy1) ;
   lisx = lisx et (prog tab3.temps. n);
 fin bou1 ;
ev1 = evol manu 'temps' lisx 'depl' lisy ;
list lisy;
 si graph ; dess ev1  ; finsi ;

* la reference pour le test est le resultat d'une
*             certaine version de C2000
dmax = maxi abs lisy ; list dmax ;
si complet ;
valtes = 1.1;
sinon;
valtes =  .082;
finsi;
err1 = abs ((dmax - valtes)/valtes);
list err1 ;
 si (err1 > .05) ; erreur 5 ;
 sinon ; erre 0 ; finsi ;
fin ;
```

Test dyna_nl3 **Comments**

1. **First step : Static computation under the compression force and the disturbing transversal force**

```
fo2 = depi cl3 .01 ;
fo0 = manu chpo geo 1 p (2050781.25*1.05) nature discret ;
*
dt = 0.00001 ;
li1 = prog 0. dt ;
li2 = prog 0. 1 ;
ev = evol manu t (li1 ) f(t) (li2 ) ;
fcons = char dimp fo2 ev ;
fncons = char psui fo0 ev ;
fco = fncons et fcons ;
lis1a = prog 0. pas (dt/5) dt ;
*
tab3 = table ;
tab3.grands_deplacements = vrai ;
tab3.chargement = fco ;
tab3.blocages_mecaniques = cl1 et cl2 et cl3 ;
tab3.modele = mototal ;
tab3.caracteristiques = mattotal ;
tab3.temps_calcules = lis1a ;
pasapas tab3 ;
```

In the static computation the free tip of the beam has an imposed vertical displacement (fo2). The follower pressure is described with the fo0 object. The loads are defined as a function of time with CHAR operator and the evolution ev.
The PASAPAS subroutine is called by using 6 computation steps (defined with lis1a).

2. **Second step : Dynamic computation under the compression force and the release of the beam**

```
ev2 = evol manu (prog dt 100) (prog 1 1 ) ;
*
fncons = char psui fo0 ev2 ;
si complet;
lis1c = prog dt pas .001 .045 ;
sinon;
lis1c = prog dt pas .001 0.005;
finsi;
tab3.chargement = fncons ;
tab3.blocages_mecaniques = cl1 et cl2 ;
tab3.dynamique = vrai ;
tab3.temps_calcules = lis1c ;
*
pasapas tab3 ;
```

The beam is released : the boundary condition cl3 and imposed displacement fo2 are removed. The applied load is the follower pressure defined as a function of time (fncons).
The computation is a dynamic analysis so tab3.dynamique is true.

Chapter 14

THERMAL ANALYSIS

14.1 Test ther2 Description sheet

Test name ther2

Calculation type THERMAL STEADY STATE
2D AXISYMMETRICAL

Finite element type QUA4

Topic
Test of the boundary conditions of temperature with an axisymmetrical computation.

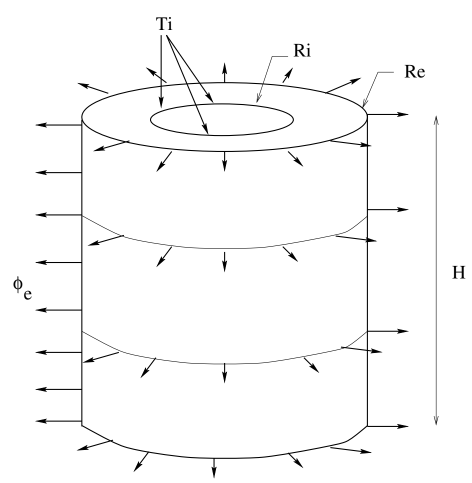
Three different configurations are tested on a thick-walled cylinder :
- an imposed temperature and an imposed flux
- an imposed temperature and a volumetric source of heat
- an imposed temperature and a forced convection

Goal Find the field of temperatures in the cylinder for each configuration.

Reference Analytical solution (1st and 2nd parts).
Comparison with results computed by THERMX (3rd part).

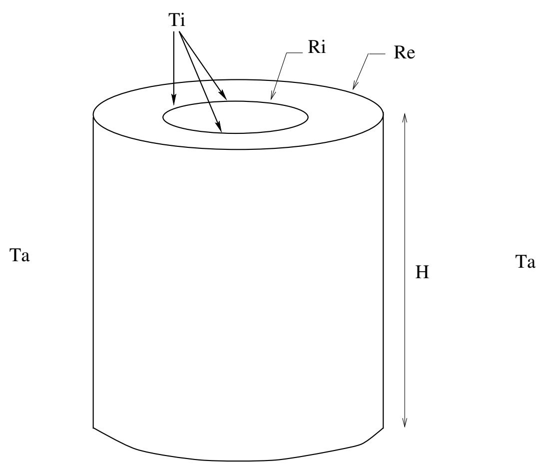
Version 97' customer version

Model description



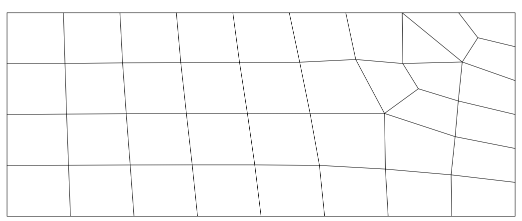
1st configuration

$R_i = 0.0515m$
$R_e = 0.2015m$
$H = 0.5m$
$T_i = 0.57^oC$
$\phi_e = 0.72W/m^2$
$K = 130W/m^o$

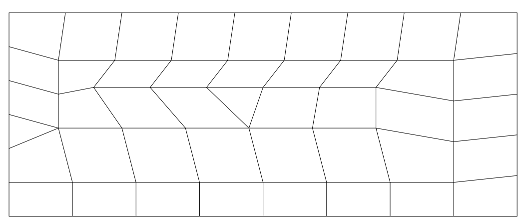


2nd configuration		3rd configuration	
R_i	$= 6m$	R_i	$= 6m$
R_e	$= 16m$	R_e	$= 16m$
H	$= 4m$	H	$= 4m$
T_i	$= 100^oC$	T_i	$= 100^oC$
ϕ_e	$= 0.W/m^2$	T_a	$= 125^oC$
K	$= 1W/m^o$	K	$= 8W/m^o$
Q	$= 20W$	H_c	$= 5W/m^{2o}$

CASTEM figures



MESHING - 2ND PROBLEM



MESHING - 3RD PROBLEM

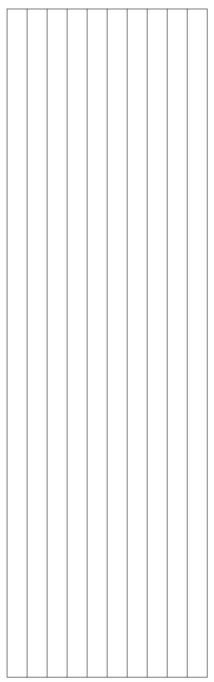
Test ther2 **Results**

Results

Value of the temperature at the point located on the external radius.

	Reference	Computed
Configuration 1	0.571	0.571
Configuration 2	1510.9	1505.3
Configuration 3	122.68	122.68

CASTEM figures



MESHING - 1ST PROBLEM

```
*          Test Ther2.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
GRAPH='N';
SAUT PAGE ;
*---------------------------------------------------*
* TEST THER2                                        *
*                                                   *
* TEST DES CL DE TEMPERATURE EN MODE AXISYMETRIQUE  *
* -----------------------------------------------   *
*                                                   *
*    TEMPERATURE IMPOSEE + CONVECTION               *
*                        + FLUX                     *
*                        + SOURCE                   *
*                                                   *
*                                                   *
* Comme dans le test THER1, cet exemple permet de   *
* tester les conditions aux limites de température en *
* calcul axisymétrique.                             *
*                                                   *
* Trois cas sont envisagés ici :                    *
*      - un tube épais soumis à une température     *
*        imposée et à un flux imposé                *
*      - un tube épais soumis à une température     *
*        imposée et à une source volumique de chaleur *
*      - un tube épais soumis à une température     *
*        imposée et à une convection forcée         *
*                                                   *
*---------------------------------------------------*
*
*--- 1er problème
*    ------------
*
*---  Description :
*
*     établir le champ de température dans un tube
*     épais soumis à une condition de température
*     imposée et à un flux imposé
*                     -----------
*---  Description de la géométrie :
*
*  |  axe de symétrie
*  .
*  |     _________________________
*  .   |                         |
*  .   |                         |
*  .   |                         |
*  .   |                         |
*  .   |                         |
*  .   |                         |
*  |   |         K=130.          |
*  .   |_________________________|   -------> R (R>0)
*  |   TI=0.57                  F=0.72
*      RI=0.0515                RE=0.2015
*
*---  Commentaires :
*
*     le problème est à symétrie de révolution.
*     température imposée sur la paroi interne (TI=0.57)
*     flux imposé sur la paroi externe (F=0.72)
*     conductibilité isotrope (K=130)
*
*---  Solution analytique :
*
*     T(R) = TI + LOG(R/RI) * (F * RE)/K
*     avec notammant :
*     T(R=0.2015) = 0.5715225
*
*-----------------------------------------------------
*
TITRE 'THER2 - C.L. TEMPERATURE - CALCUL AXISYMETRIQUE';
OPTION DIME 2 ELEM QUA4 MODE AXIS ;
OPTION ECHO 0 ;
TEMPS ;
*
*---  CREATION DE LA GEOMETRIE:
*
P1 = 0.0515 0. ;P20 = 0.0665 0. ;
P19 = 0.0665 0.5 ;P2 = 0.0515 0.5 ;
P18 = 0.0815 0. ;P17 = 0.0815 0.5 ;
P16 = 0.0965 0. ;P15 = 0.0965 0.5 ;
P14 = 0.1115 0. ;P13 = 0.1115 0.5 ;
P12 = 0.1265 0. ;P11 = 0.1265 0.5 ;
P10 = 0.1415 0. ;P9 = 0.1415 0.5 ;
P8 = 0.1565 0. ;P7 = 0.1565 0.5 ;
P6 = 0.1715 0. ;P5 = 0.1715 0.5 ;
P24 = 0.1865 0. ;P23 = 0.1865 0.5 ;
P4 = 0.2015 0.5 ;P3 = 0.2015 0. ;
*
SURF1  = MANUEL QUA4 P1 P20 P19 P2 ;
SURF2  = MANUEL QUA4 P20 P18 P17 P19 ;
SURF3  = MANUEL QUA4 P18 P16 P15 P17 ;
SURF4  = MANUEL QUA4 P16 P14 P13 P15 ;
SURF5  = MANUEL QUA4 P14 P12 P11 P13 ;
SURF6  = MANUEL QUA4 P12 P10 P9 P11 ;
SURF7  = MANUEL QUA4 P10 P8 P7 P9 ;
SURF8  = MANUEL QUA4 P8 P6 P5 P7 ;
SURF9  = MANUEL QUA4 P6 P24 P23 P5 ;
SURF10 = MANUEL QUA4 P24 P3 P4 P23 ;
SURF11 = SURF1 ET SURF2 ET SURF3 ET SURF4 ET SURF5 ET
         SURF6 ET SURF7 ET SURF8 ET SURF9 ET SURF10 ;
*
SI(NEG GRAPH 'N');
    TRAC SURF11 ;
FINSI;
*
D1 = DROITE 1 P1 P2 ;
D3 = DROITE 1 P3 P4 ;
*
*---  DONNEES DU PROBLEME DE THERMIQUE:
*
*---  MODELISATION:
*
MOD1 = MODL SURF11 THERMIQUE ISOTROPE ;
*
*---  CARACTERISTIQUES DU MATERIAU:
*
MAT1 = MATR MOD1 K 130. ;
*
*---  MATRICE DE CONDUCTIVITE:
*
CND1 = CONDUCTIVITE MOD1 MAT1 ;
*
*---  TEMPERATURES IMPOSEES: BLOQUE + DEPI
*
B1 = BLOQUE T D1 ;
E1 = DEPI B1 0.57 ;
*
*---  FLUX IMPOSE:
*
F1 = FLUX MOD1 0.72 D3 ;
*
*---  ASSEMBLAGE DES PREMIER ET SECOND MEMBRES:
*
RIG1 = CND1 ET B1 ;
FF1 = E1 ET F1 ;
```

```
*
*---  RESOLUTION:
*
CHTER1 = RESOUDRE RIG1 FF1 ;
TETA1 = EXTR CHTER1 T P3 ;
*
*-------------------------------------------------------
*---  - Deuxième partie - description du problème :
*
*     établir le champ de température dans un tube
*     épais soumis à une condition de température
*     imposée ainsi qu'à une source volumique
*                           ----------------
*---  Description de la géométrie :
*
*  |  Y (axe de sym{trie)
*  .
*  |    ------------------------------
*  .   |                              |
*  |   |           Q=20.   K=1.       |
*  .    ------------------------------    -------> X
*  |   TI=100.                       F=0.
*      RI=6.                         RE=16.
*
*
*---  Commentaires :
*
*     Le problème est à symétrie de révolution.
*     Température sur la paroi interne (TI=100)
*     Flux imposé sur la paroi externe (F=0)
*                                     ( DT / DN = 0 )
*     Source volumique imposée (Q=20)
*     Conductibilité isotrope (K=1)
*
*--- Solution analytique :
*
*     T(R) = TI - (Q/4K)*(R**2-RI**2) +
*            (Q*RE/2)*RE*LOG(R/RI)
*     avec notamment :
*     T(R=16) = 1510.923
*
*-------------------------------------------------------
*
*---  CREATION DE LA GEOMETRIE:
*
P1 = 6 0 ;
P2 = 16 0 ;
P3 = 16 4 ;
P4 = 6 4 ;
D1 = DROITE 8 P1 P2 ;
D2 = DROITE 6 P2 P3 ;
D3 = DROITE 9 P3 P4 ;
D4 = DROITE 4 P4 P1 ;
LIG1 = D1 ET D2 ET D3 ET D4 ;
SURF1 = SURFACE LIG1 PLANE ;
*
SI(NEG GRAPH 'N');
   TRAC SURF1 ;
FINSI;
*
*
*---  DONNEES DU PROBLEME DE THERMIQUE:
*
*---  MODELISATION:
*
MOD1 = MODL SURF1 THERMIQUE ISOTROPE ;
*
*---  CARACTERISTIQUES DU MATERIAU:
*
MAT1 = MATR MOD1 K 1. ;
*
*---  MATRICE DE CONDUCTIVITE:
*
CND1 = CONDUCTIVITE MOD1 MAT1 ;
*
*---  TEMPERATURES IMPOSEES: BLOQUE + DEPI
*
B1 = BLOQUE T D4 ;
E1 = DEPI B1 100. ;
*
*---  FLUX IMPOSE:
*
E2 = FLUX MOD1 0. D2 ;
*
*---  SOURCE VOLUMIQUE:
*
E3 = SOURCE MOD1 20. SURF1 ;
*
*---  ASSEMBLAGE DES PREMIER ET SECOND MEMBRE:
*
RIG1 = CND1 ET B1 ;
FF1 = E1 ET E2 ET E3 ;
*
*---  RESOLUTION:
*
CHTER1 = RESOUDRE RIG1 FF1 ;
TETA2 = EXTR CHTER1 T P3 ;
*
*
*-------------------------------------------------------
*
*---  Troisième partie
*
*---  Description du problème :
*
*     établir le champ de température dans un tube
*     épais soumis à une condition interne de tempéra-
*     ture ainsi qu'à une convection sur sa paroi
*     externe
*
*---  Données du problème :
*
*     Conductibilité isotrope du matériau (K=8)
*     Coefficient d'échange en convection (HC=5)
*     Température imposée sur la paroi interne (TI=100)
*     Température du milieu extérieur (TE=125)
*
*---  Solution :
*
*     On pourra comparer avec les résultats suivants
*     issus d'un calcul effectué par THERMX :
*
*     100.00 / 104.37 / 108.05 / 111.22 / 114.01 /
*     116.50 / 118.75 / 120.80 / 122.68
*     (valeurs du champ thermique selon l'abscisse)
*
*-------------------------------------------------------
*
*---  CREATION DE LA GEOMETRIE:
*
*     - rayon int{reiur RI = 6.
*     - rayon ext{rieur RE = 16.
*     - hauteur         H  = 4.
*     - surface interne : D4
*     - surface externe : D2
*
```

```
P1 = 6 0 ;
P2 = 16 0 ;
P3 = 16 4 ;
P4 = 6 4 ;
D1 = DROITE 8 P1 P2 ;
D2 = DROITE 5 P2 P3 ;
D3 = DROITE 9 P3 P4 ;
D4 = DROITE 6 P4 P1 ;
LIG1 = D1 ET D2 ET D3 ET D4 ;
SURF1 = SURF LIG1 PLANE ;
*
SI(NEG GRAPH 'N');
    TRAC SURF1 ;
FINSI;
*
*
*---  DONNEES DU PROBLEME DE THERMIQUE:
*
*--- MODELISATION:
*
MOD1 = MODL SURF1 THERMIQUE ISOTROPE ;
MOD2 = MODL D2 CONVECTION ;
*
*---  DONNEE DES CARACTERISTIQUES DU MATERIAU:
*
MAT1 = MATR MOD1 K 8. ;
MAT2 = MATR MOD2 H 5. ;
*
*---  MATRICES DE CONDUCTIVITE:
*
CND1 = CONDUCTIVITE MOD1 MAT1 ;
CNF1 = CONDUCTIVITE MOD2 MAT2 ;
*
*---  FLUX EQUIVALENTS A LA CONVECTION:
*
FF1 = CONVECTION MOD2 MAT2 T 125. ;
*
*---  TEMPERATURE IMPOSEE: BLOQUE + DEPI
*
B1 = BLOQUE T D4 ;
E1 = DEPI B1 100. ;
*
*---  ASSEMBLAGE DU PREMIER PUIS DU SECOND MEMBRE:
*
RIG1 = CND1 ET CNF1 ET B1 ;
Q1 = FF1 ET E1 ;
*
*---  RESOLUTION:
*
CHTER1 = RESOUDRE RIG1 Q1 ;
TETA3 = EXTR CHTER1 T P3 ;
*
*--- POST-TRAITEMENT
*
SAUT PAGE ;
SAUT 2 LIGN ;
MESS ' PROBLEME N1 ' ;
MESS ' TEMPERATURE EN R = RE         THEORIQUE       CALCULEE ';
MESS '                              T = 0.5715225     T = ' TETA1 ;
SAUT 1 LIGN ;
MESS ' PROBLEME N2 ' ;
MESS ' TEMPERATURE EN R = RE         THEORIQUE       CALCULEE ';
MESS '                              T = 1510.923      T = ' TETA2 ;
SAUT 1 LIGN ;
MESS ' PROBLEME N3 ' ;
MESS ' TEMPERATURE EN R = RE         REFERENCE       CALCULEE ';
MESS '                              T = 122.68        T = ' TETA3 ;
*
TEMPS ;
*        CODE DE FONCTIONNEMENT SUR LE PROBLEME N2
TREF=1511;RESI=ABS((TETA2-TREF)/TREF);
SI(RESI <EG 5E-2);
   ERRE  0;
SINO;
   ERRE  5;
FINSI;
FIN;
```

Test ther2 **Comments**

1. **First problem**

   - Definition of the thermic model

     MOD1 = MODEL SURF11 THERMIQUE ISOTROPE ;
     MAT1 = MATER MOD1 K 130. ;
     CND1 = CONDUCTIVITE MOD1 MAT1 ;

     The thermic model (MOD1) is defined with the formulation named THERMIQUE ISOTROPE. This model is described by the geometry SURF11, the conductivity of the model is defined with the operator MATER.
     The operator COND(UCTIVITE) builds the matrix of conductivity (CND1).

   - Definition of the thermic boundary conditions

     B1 = BLOQUE T D1 ;
     E1 = DEPI B1 0.57 ;
     F1 = FLUX MOD1 ;
     The operator BLOQUE builds the boundary condition (B1) on the degree of freedom T of the line D1. The operator DEPI(MPOSE) imposes a value of 0.57 on the condition B1.
     The operator FLUX imposes a flux of 0.72 on the line D3 which belongs to MOD1.

   - Computation

     The first and second members of the equation to be solved are gathered and the computation is performed with the operator RESO. The computed object CHTER1 is a CHPOINT type and the values of its component T can be extracted with the operator EXTR(AIRE).

2. **Second problem**

   E3 = SOURCE MOD1 20. SURF1 ;

   The model, the imposed temperature and the flux are defined in the same way as in the first configuration.
   The operator SOURCE defines an imposed volumetric source of heat on the surface SURF1.
   The computation is performed like in the first problem by gathering first and second members of the equation and then by solving with the RESO operator.

3. **Third problem**

   MOD1 = MODEL SURF1 THERMIQUE ISOTROPE ;
   MOD2 = MODEL D2 CONVECTION ;
   *
   MAT1 = MATER MOD1 K 8. ;
   MAT2 = MATER MOD2 H 5. ;
   *
   CND1 = CONDUCTIVITE MOD1 MAT1 ;
   CNF1 = CONDUCTIVITE MOD2 MAT2 ;
   *
   FF1 = CONVECTION MOD2 MAT2 T 125. ;

   The outer surface is submitted to forced convection : as the problem is axisymmetrical, the outer surface is modelled by the line D2. Along this line is defined a convection model (MOD2) by using the operator MODEL followed by the formulation CONVECTION.
   The operator MATER defines the convection exchange coefficient.
   The operator CONV(ECTION) imposes a condition of convection on the structure defined in the objects MOD2 and MAT2, it is necessary to indicate the value of the external temperature (T=125).

14.2 Test ther7or Description sheet

Test name ther7or

Calculation type THERMAL STEADY STATE
3D ORTHOTROPIC

Finite element type COQ8 COQ6

Topic
Test of the boundary conditions of temperature with orthotropic elements.

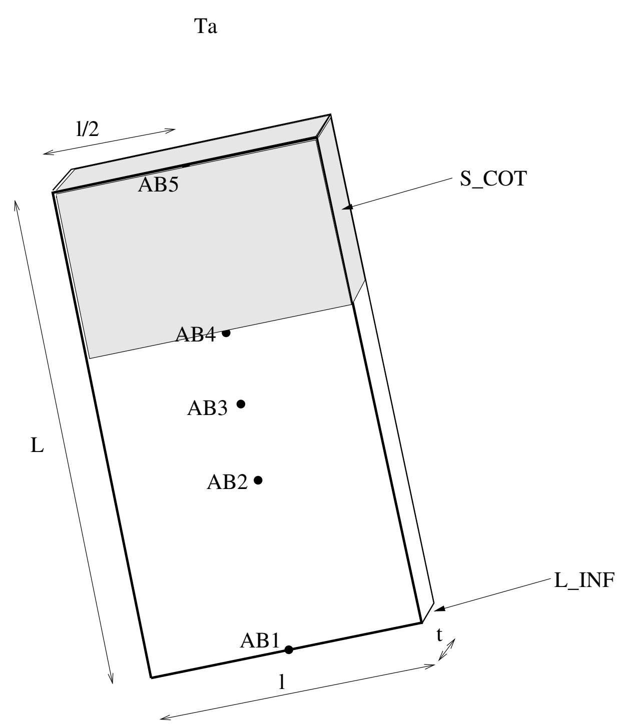
Three different configurations are tested on a shell :
- an imposed temperature and a forced convection
- an imposed temperature and an imposed flux
- an imposed temperature and a volumetric source of heat

Goal Find the field of temperatures in the shell for each configuration and compare it with the case of isotropic elements.

Reference CASTEM Dowlatyari (03/91).

Version 97' customer version

Model description

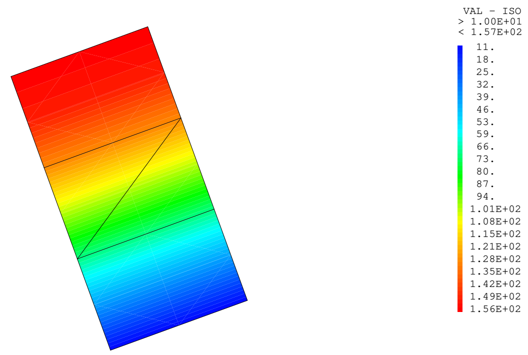


$L = 6m$
$l = 3m$
$t = 0.5m$
$K_1 = 100W/m^o$
$K_2 = 1000W/m^o$
$K_3 = 100W/m^o$
$T_{L\_inf} = 10^oC$
$H_{S\_COT} = 15W/m^{2o}$
$T_a = 200^oC$
$\phi_{S\_COT} = 200W/m^2$
$Q_{S\_COT} = 1000W/m^3$

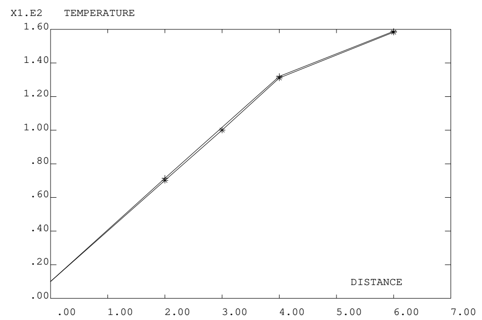
Test ther7or **Results**

CASTEM figures

1st resolution : convection

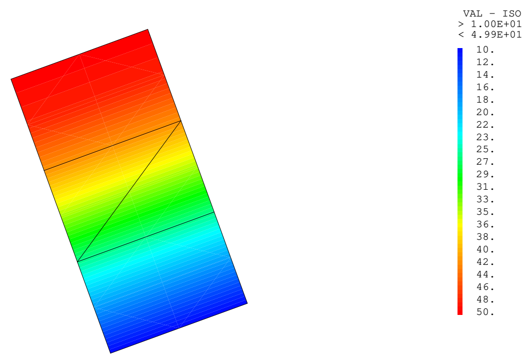


ELEMENTS COQ6,COQ8 : CONVECTION

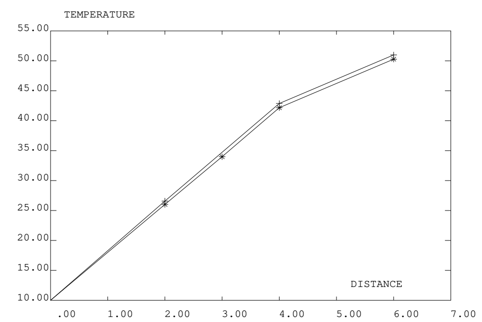


TEMPERATURE + CONVECTION. + :ISOTROPIC * :ORTHOT.

2nd resolution : imposed flux

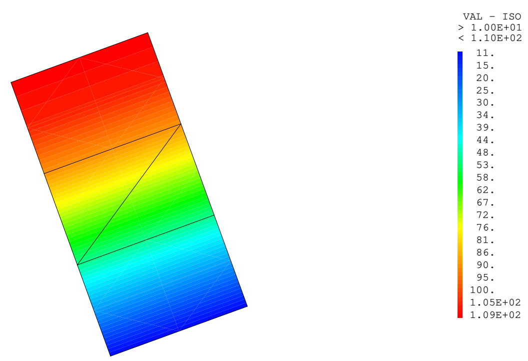


ELEMENTS COQ6,COQ8 : IMPOSED FLUX

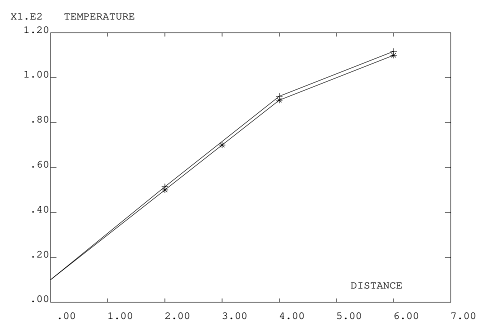


TEMPERATURE + IMPOSED FLUX. + :ISOTROPIC * :ORTHOT.

3rd resolution : volumetric source



ELEMENTS COQ6,COQ8 : SOURCE



TEMPERATURE + SOURCE. + :ISOTROPIC * :ORTHOT.

```
*         Test Ther7or.dgibi: Jeux de données        *
*         -----------------------------------        *
*                                                    *
SAUT PAGE ;
*----------------------------------------------------*
* TEST THER7OR                                       *
*                                                    *
*    TEST DES CL DE TEMPERATURE POUR COQ8 ET COQ6    *
*                   ORTHOTROPE                       *
*    -------------------------------------------     *
*                                                    *
*               TEMPERATURE IMPOSEE + CONVECTION     *
*                                   + FLUX           *
*                                   + SOURCE         *
*                                                    *
* Ce test permet de vérifier le bon fonctionnement   *
* des divers opérateurs thermiques de CASTEM2000 en  *
* ORTHOTROPIE                                        *
*                                                    *
* UNE PLAQUE RECTANGULAIRE CONSTITUEE D'éléments COQ8 *
* et coq6 est soumise à une température imposée à une *
* de ses extrémités et sucessivement à une condition  *
* de convection forcée, à une condition de flux impo- *
* -sé et à une condition de source volumique imposée. *
*                                                    *
*                                                    *
* Il s'agit du meme probleme que pour le test ther71. *
* Mais on utilise un materiau orthotrope au lieu d'un *
* materiau isotrope avec la meme conductivite suivant *
* l'axe A1A7. Or le gradient de temperature etant    *
* parallele a cet axe, on doit obtenir un resultat   *
* identique.                                         *
* On compare donc les resultats obtenus a ceux du    *
* test ther71 pour les elements coques.              *
*                                                    *
* Test elabore par Mr Dowlatyari 03/91.              *
* Test modifie pour evolution 21/05/92.              *
*                                                    *
*----------------------------------------------------*
opti echo 1;
opti trac bens;
graph = 'N';
SAUT PAGE ;
*
TITRE 'C.L. DE TEMPERATURE - ELEMENTS  COQ8 , COQ6 ' ;
OPTION DIME 3 ELEM QUA8 ;
TEMPS ;
*
*CREATION DE LA GEOMETRIE: POINTS SUPPORTS DES ELEMENTS
*
A1 = 0. 0. 0. ; B1 = 0. 3.0 0. ;AB1 =0. 1.5  0.  ;
A2= 0. 0. 1. ;  B2 = 0. 3. 1. ; AB2 = 0. 1.5 2.  ;
A3 = 0. 0. 2. ; B3 = 0. 3. 2. ; AB4 = 0. 1.5 4.  ;
A4 = 0. 0. 3. ; B4 = 0. 3. 3. ; AB5 = 0. 1.5 6.  ;
A5 = 0. 0. 4. ;B5 = 0. 3. 4. ; AB3= 0. 1.5 3.  ;
A6 = 0. 0. 5. ;B6 = 0. 3. 5. ;
A7 = 0. 0. 6. ;B7 = 0. 3. 6. ;
M=0. 0. 0. ;N=2. 0. 0. ;
*
ANG=20.0  ;
A1=A1 TOUR ANG M N ;A2=A2 TOUR ANG M N ;
A3=A3 TOUR ANG M N ;A4=A4 TOUR ANG M N ;
A5=A5 TOUR ANG M N ;A6=A6 TOUR ANG M N ;
A7=A7 TOUR ANG M N ;B1=B1 TOUR ANG M N ;
B2=B2 TOUR ANG M N ;B3=B3 TOUR ANG M N ;
B4=B4 TOUR ANG M N ;B5=B5 TOUR ANG M N ;
B6=B6 TOUR ANG M N ;B7=B7 TOUR ANG M N ;
AB1=AB1 TOUR ANG M N ;AB2=AB2 TOUR ANG M N ;
AB3=AB3 TOUR ANG M N ;AB4=AB4 TOUR ANG M N ;
AB5=AB5 TOUR ANG M N ;
*------- CREATION DES QUADRANGLES ET TRIANGLES -------
*
QH01 = MANU QUA8 A1 AB1 B1 B2 B3 AB2  A3 A2   ;
QH02 = MANU TRI6 A3 AB2 B3 B4 B5 AB3      ;
QH03 = MANU TRI6 A3 AB3 B5 AB4 A5 A4      ;
QH04 = MANU QUA8 A5  AB4 B5 B6  B7 AB5  A7 A6 ;
*
*
QTOT = QH01 ET QH02 ET QH03 ET QH04 ;

si ( NEG GRAPH 'N' );
OEIL = 100. 100. 100. ;
TRACE OEIL QTOT;
finsi;
QSOUR = QH04 ;
*
*
L_INF = MANU SEG3 A1 AB1 B1 ;
*
S_COT = QH04;
*
* ATTENTION : il faut réorienter les éléments
* (OPERATEUR ORIENT) si on n'est pas sur que tous les
* éléments sont orientés d'une façon cohérante
*
*----------- DONNEES DU PROBLEME DE THERMIQUE --------
*
*--------------  MODELISATION ------------------------
*
MOD1 = MODL QTOT THERMIQUE ORTHOTROPE COQ8 COQ6 ;
MOD2 = MODL S_COT CONVECTION SUPERIEURE COQ8 COQ6 ;
*
*------ DONNEES DES CARACTERISTIQUES DU MATERIAU ------
*
MAT1 = MATR MOD1 'DIRECTION' A2 'PARALLELE'
'K1' 100. 'K2' 1000.  'K3' 100. EPAI 0.5    ;
MAT2 = MATR MOD2 'H' 15. ;
*
*------- CREATION DES MATRICES DE CONDUCTIVITE --------
*
*OPTION DONN 5 ;

CND1 = CONDUCTIVITE MOD1 MAT1 ;
CND2 = CONDUCTIVITE MOD2 MAT2 ;
*
*-------- TEMPERATURES IMPOSEES: BLOQUE + DEPI --------
*
BB1 = BLOQUE L_INF 'TINF' ;
EE1 = DEPI BB1 10. ;
BB2 = BLOQUE L_INF 'TSUP' ;
EE2 = DEPI BB2 10. ;
BB3 = BLOQUE L_INF 'T   ' ;
EE3 = DEPI BB3 10. ;
BB  = BB1 ET BB2 ET BB3 ;

EE  = EE1 ET EE2 ET EE3 ;

*
*--- FLUX EQUIVALENTS A LA CONDITION DE CONVECTION ----
*
CNV1 = CONVECTION MOD2 MAT2 'T' 200.;
*
*--------------- FLUX IMPOSE --------------------------
*
FLU1 = FLUX MOD1 200.  S_COT SUPE ;
*
```

```
*--------------- SOURCE DE CHALEUR --------------------
*
list qsour;
S1 = SOURCE MOD1 MAT1  1000. QSOUR ;
list s1;
*
*-------- ASSEMBLAGE DES TERMES DE CONDUCTIVITE -------
*
CCC1 = CND1 ET CND2 ET  BB  ;CCC2 = CND1 ET BB ;
*
*------ ASSEMBLAGE DES TERMES DE FLUX EQUIVALENTS -----
*
FFF1 = EE ET CNV1 ;FFF2 = EE ET FLU1 ;FFF3 = EE ET S1 ;
*FFF1 = EE ET CNV1  ;FFF2 = EE ET FLU1 ;
*
*----------- 1ERE RESOLUTION : CONVECTION -------------
*
*OPTION DONN 5 ;
CHTER1 = RESOUDRE CCC1 FFF1 ;
*
*---------------- 2EME RESOLUTION : FLUX --------------
*
CHTER2 = RESOUDRE CCC2 FFF2 ;
*
*---------------- 3EME RESOLUTION: SOURCE -------------
*
CHTER3 = RESOUDRE CCC2 FFF3 ;
*
*----- POST-TRAITEMENT: TRACE DES CHAMPS RESULTATS ----
*------ ET EXTRACTION DES VALEURS CARACTERISTIQUES ----
*
*LISTE CHTER1 ;
*LISTE CHTER2 ;
*LISTE CHTER3 ;
*
OEIL= 1000. 0. 0. ;
*
SI (NEG GRAPH 'N');
TITRE 'ELEMENTS COQ6,COQ8 : CONVECTION' ;
CHTER1I = 'EXCO' CHTER1 'TINF' 'NOID' 'TINF' ;
TRACER OEIL QTOT CHTER1I ;
TITRE 'ELEMENTS COQ6,COQ8 : FLUX IMPOSE' ;
CHTER2I = 'EXCO' CHTER2 'TINF' 'NOID' 'TINF' ;
TRACER OEIL QTOT CHTER2I ;
TITRE 'ELEMENTS COQ6,COQ8  : SOURCE' ;
CHTER3I = 'EXCO' CHTER3 'TINF' 'NOID' 'TINF' ;
TRACER OEIL QTOT CHTER3I ;
FINSI;

T1AB1 = EXTR CHTER1 TSUP AB1 ;
T1AB2=EXTR CHTER1 TSUP AB2 ;
T1AB3 = EXTR CHTER1 TSUP AB3 ;
T1AB4=EXTR CHTER1 TSUP AB4 ;
T1AB5 = EXTR CHTER1 TSUP AB5 ;
T2AB1= EXTR CHTER2 TSUP AB1 ;
T2AB2=EXTR CHTER2 TSUP AB2 ;
T2AB3 = EXTR CHTER2 TSUP AB3 ;
T2AB4=EXTR CHTER2 TSUP AB4 ;
T2AB5 = EXTR CHTER2 TSUP AB5 ;
T3AB1 = EXTR CHTER3 TSUP AB1 ;
T3AB2=EXTR CHTER3 TSUP AB2 ;
T3AB3 = EXTR CHTER3 TSUP AB3 ;
T3AB4=EXTR CHTER3 TSUP AB4 ;
T3AB5 = EXTR CHTER3 TSUP AB5 ;
*

SI ( NEG GRAPH 'N') ;
DIST = PROG 0. 2. 4. 6. ;
DIST1 = PROG 0. 2. 3. 4. 6. ;
TEMPR1=PROG 10. 71.44 132.03 158.86 ;
TEMPC1= PROG T1AB1 T1AB2 T1AB3 T1AB4 T1AB5 ;
EV1R = EVOL MANU 'DISTANCE' DIST 'TEMPERATURE' TEMPR1 ;
EV1C = EVOL MANU 'DISTANCE' DIST1 'TEMPERATURE' TEMPC1;
TAB = 'TABLE' ;
TAB.1 = 'MARQ PLUS' ;
TAB.2 = 'MARQ ETOI' ;
TITRE 'COQ6 ET COQ8 ;TEMPERATURE + CONVECTION ;+ :
MASSIF,* : COQUE' ;
DESSIN TAB (EV1R ET EV1C )  ;
TEMPR2=PROG 10. 26.58 42.878 50.984 ;
TEMPC2= PROG T2AB1 T2AB2 T2AB3 T2AB4 T2AB5 ;
EV2R = EVOL MANU 'DISTANCE' DIST 'TEMPERATURE' TEMPR2 ;
EV2C = EVOL MANU 'DISTANCE' DIST1 'TEMPERATURE' TEMPC2;
TITRE 'COQ6 ET COQ8;TEMPERATURE + FLUX IMPOSE ;+ :
MASSIF,* : COQUE';
DESSIN TAB (EV2R ET EV2C )  ;
TEMPR3=PROG 10. 51.43 91.75  111.75   ;
TEMPC3= PROG T3AB1 T3AB2 T3AB3 T3AB4  T3AB5 ;
EV3R = EVOL MANU 'DISTANCE' DIST 'TEMPERATURE' TEMPR3 ;
EV3C = EVOL MANU 'DISTANCE' DIST1 'TEMPERATURE' TEMPC3;
TITRE 'COQ6 ET COQ8 ;TEMPERATURE + SOURCE ;+
MASSIF,* : COQUE ';
DESSIN TAB (EV3R ET EV3C )  ;
TEMPS ;
FINSI;
T1Aq = 10.; T1Bq = 71.44; t1Cq = 132.03; T1DQ = 158.86;
T2Aq = 10.; T2Bq = 26.58; t2Cq = 42.878; T2DQ = 50.984;
T3AQ = 10.; T3BQ = 58.38; T3CQ = 105.34 ; T3DQ = 123.33;


********************************************************
**************  TEST DE FONCTIONNEMENT ****************
********************************************************
mess t1ab1  T1Aq T1AB2 T1Bq;
mess T1AB4 T1Cq T1AB5 T1Dq;
mess T2AB1 T2Aq T2AB2 T2Bq;
mess T2AB4 T2Cq T2AB5 T2Dq;
mess T3AB1 T3Aq T3AB2 T3Bq;
mess T3AB4 T3Cq T3AB5 T3Dq;
ERCA= ABS (( T1AB1-T1Aq)/T1AB1);
ERCB= ABS (( T1AB2-T1Bq)/T1AB2);
ERCC= ABS (( T1AB4-T1Cq)/T1AB4);
ERCD= ABS (( T1AB5-T1Dq)/T1AB5);
ERFA= ABS (( T2AB1-T2Aq)/T2AB1);
ERFB= ABS (( T2AB2-T2Bq)/T2AB2);
ERFC= ABS (( T2AB4-T2Cq)/T2AB4);
ERFD= ABS (( T2AB5-T2Dq)/T2AB5);
ERSA= ABS (( T3AB1-T3Aq)/T3AB1);
ERSB= ABS (( T3AB2-T3Bq)/T3AB2);
ERSC= ABS (( T3AB4-T3Cq)/T3AB4);
ERSD= ABS (( T3AB5-T3Dq)/T3AB5);

ERTOT = PROG ERCA ERCB ERCC ERCD ERFA ERFB ERFC ERFD
ERSA ERSB ERSC ERSD;
ermax = maxi ertot;
temps;
SI ( ERMAX <EG 0.05 );
   ERRE 0;
SINON;
   ERRE 5;
FINSI;
fin;
```

Test ther7or **Comments**

1. **Orthotropic definition**

   MOD1 = MODEL QTOT THERMIQUE ORTHOTROPE COQ8 COQ6 ;
   MAT1 = MATER MOD1 'DIRECTION' A2 'PARALLELE'
   'K1' 100. 'K2' 1000. 'K3' 100. EPAI 0.5 ;

   The thermal model (MOD1) is defined with the formulation named THERRMIQUE ORTHOTROPE. This model is described by the geometry QTOT, the conductivity of the model is defined with the operator MATER : K1, K2 and K3 are the thermal conductivities and the thickness is equal to 0.5 ; the first orthotropic direction is parallel to the vector defined by A2.

2. **Model of convection**

   MOD2 = MODEL S_COT CONVECTION SUPERIEURE COQ8 COQ6 ;
   MAT2 = MATER MOD2 'H' 15. ;

   The model MOD2 defines a convection formulation. The keyword SUPERIEURE indicates that the convection is located on the upper side of the shell.

3. **Degrees of freedom**

   BB1 = BLOQUE L_INF 'TINF' ;
   BB2 = BLOQUE L_INF 'TSUP' ;
   BB3 = BLOQUE L_INF 'T ' ;

   The shells have 3 degrees of freedom : the inside temperature T, the upper temperature TSUP and the lower temperature TINF.

14.3 Test ther-perm Description sheet

Test name ther-perm

Calculation type THERMAL STEADY STATE
3D NON LINEAR

Finite element type QUA4

Topic
Use of the procedure THERMIC in a non linear steady state.

An endless shell is submitted to a volumetric source of heat and to an imposed temperature on one side. The conductivity is a linear function of the temperature.

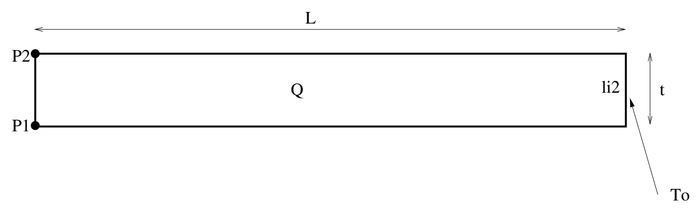
Goal Find the field of temperatures in the shell and compare it with the analytical solution.

Reference

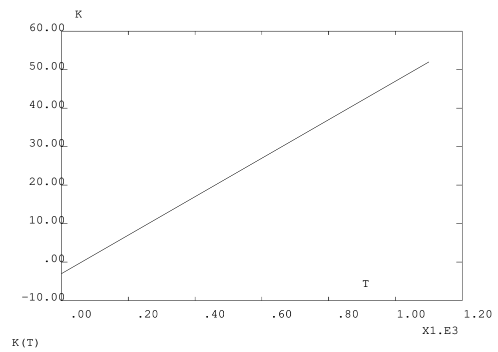
- Klaus-Jürgen Bathe & Mohammad R. Khoshgoftaar, Finite element formulation and solution of non-linear heat transfer, Nuclear Engineering and Design, v. 51 (1979), pp. 389-401
- J. Joly, Cas tests non linéaires de validation pour DELFINE, Note technique EMT.SMTS.TTMF 84/29
- V. Arpaci, Conduction Heat Transfer, Adison-Wesley, 1966, pp. 130-132

Version 97' customer version

Model description

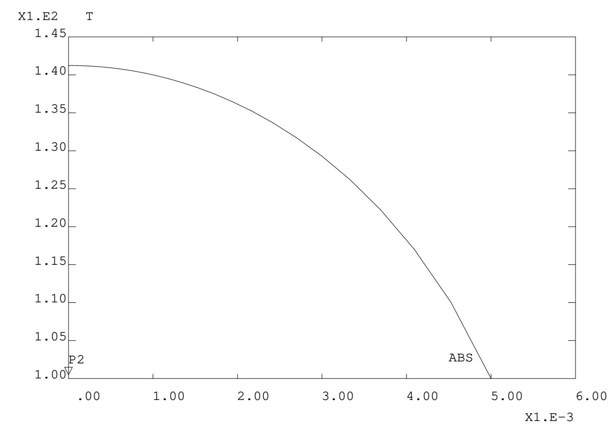


$$
\begin{aligned}
L &= 5e-3m \\
t &= L/10 \\
T_{li2} &= 100^oC \\
T_o &= 100^oC \\
Q &= 10^7 W/m^3 \\
K(T) &= 0.05 * T - 3
\end{aligned}
$$

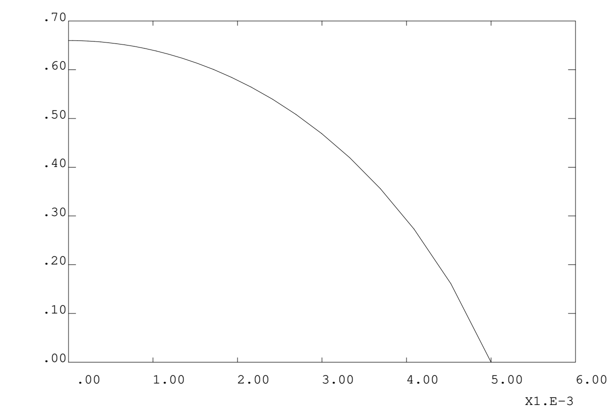


Test ther-perm
CASTEM figures

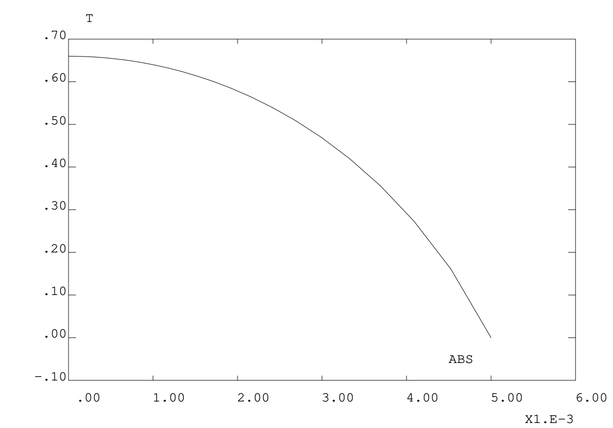
Results



Profile of absolute temperature



Profile of relative temperature (analytic solution)



Profile of relative temperature (Finite Element solution)

```
*       Test Ther-perm.dgibi: Jeux de données       *
*       ------------------------------------        *
*                                                   *
*****************************************************
*                                                   *
*   Calcul d'une plaque infinie (de largeur 2L) avec  *
*   une source volumique et température imposéé sur   *
*   les bords. Conductivité dépend linéairement de la *
*   température.                                    *
*                                                   *
*   Modélisation plane.                             *
*                                                   *
*   Auteur : Michel Bulik                           *
*                                                   *
*   Date : Décembre 1996                            *
*                                                   *
*   Références :                                    *
*   -----------                                     *
*   [1] Klaus-Jürgen Bathe & Mohammad R. Khoshgoftaar,*
*       Finite element formulation and solution of   *
*       non-linear heat transfer, Nuclear Engineering *
*       and Design, v. 51 (1979), pp. 389-401        *
*                                                   *
*   [2] J. Joly, Cas tests non linéaires de validation*
*       pour DELFINE, Note technique EMT.SMTS.TTMF   *
*       84/29                                        *
*                                                   *
*   [3] V. Arpaci, Conduction Heat Transfer, Adison-  *
*       Wesley, 1966, pp. 130-132                    *
*                                                   *
*****************************************************
*                                                   *
*  Les résultats du calcul sont comparés avec la     *
*  solution analytique (voir [3]).                   *
*                                                   *
*****************************************************

*** Options ...

    opti dime 2 elem seg2 echo 1 ;

*** Paramètres ...

    L = 5.e-3 ;
    ep = L / 10 ;

    K0 = 2. ;
    beta = 0.025 ;

    T0 = 100. ;

    q = 10000000. ;

    graph = faux ;

*** Points ...

    dens ep ;
    p1 = 0 0 ;
    p2 = 0 ep ;

    vechoriz = L 0 ;

*** Lignes ...

    li1 = p1 d 1 p2 ;

*** Surface ...

    opti elem qua4 ;
    su1 = li1 tran vechoriz dini (L/100.) dfin (L/10.);

    li2 = cote 3 su1 ;

    li3 = cote 2 su1 ;

    si(graph) ;
       titr 'Le maillage de la plaque' ;
       trac su1 ;
    finsi ;

*** Modèles ...

    mocnd = mode su1 thermique ;

    dt1 = -100. ;
    T1 = T0 + dt1 ;
    K1 = K0 * (beta*dt1 + 1) ;
    dt2 =  1000. ;
    T2 = T0 + dt2 ;
    K2 = K0 * (beta*dt2 + 1) ;
    titr 'K(T)' ;
    evk = evol manu 'T' (prog T1 T2) 'K' (prog K1 K2) ;
    si(graph) ;
       dess evk ;
    finsi ;

    macnd = mate mocnd 'K' evk ;

*** Température imposée ...

    blt = bloq T li2 ;

    ti1 = depi blt T0 ;

*** Source volumique ...

    fl1 = sour mocnd su1 q ;

*** Préparation de table pour THERMIC ...

    tabth = table THERMIQUE ;
    tabth . 'BLOCAGE' = blt ;
    tabth . 'IMPOSE'  = ti1 ;
    tabth . 'FLUX'    = fl1 ;
    tabth . 'INSTANT(0)' = manu chpo su1 1 T T0
                           nature diffus ;
    tabth . 'TABCOND' = table ;
    tabth . 'TABCOND' . mocnd = evk ;
    tabth . 'NIVEAU' = 1 ;

*** Lancement du calcul ...

    thermic tabth NONLINEAIRE ;

*** Post-traitement ...

    tresu = tabth . TEMPERATURE ;

    titr 'Profil de temperature absolue' ;
    evt = evol chpo tresu T li3 ;
    si(graph) ;
       dess evt ;
    finsi ;

    titr
```

```
'Profil de temperature relative (solution analytique)';
    chx = coor 1 li3 ;
    evx = evol chpo chx SCAL li3 ;
    lrx =  extr evx ABSC ;
    lr1 = prog (dime lrx) * 1. ;
    ev1 = evol manu lrx lr1 ;
*   toto1 = ev1 - ((evx / L)*(evx / L)) ;
*   toto1 = ev1 - (extr ((evx / L)*(evx / L)) cour 1) ;
    ev1x = extr (evx / L) cour 1;
    toto1 = ev1 - (ev1x * ev1x) ;
    toto2 = (toto1 * (L*L*q*beta/K0)) + ev1 ;
    kkabsc = extr toto2 ABSC ;
    kkordo = extr toto2 ORDO ;
    toto3 = evol manu kkabsc (kkordo ** 0.5) ;
    toto4 = toto3 - ev1 ;
    toto5 = (toto4 * (2*K0/(beta*q*L*L))) coul VERT ;

    si(graph) ;
       dess toto5 ;
    finsi ;

    titr
'Profil de temperature relative (solution EF)' ;
    evtrel = ((evt - (T0 * ev1)) * (2*K0/(q*L*L)))
             coul ROUG ;
    si(graph) ;
       dess (evtrel et toto5) ;
    finsi ;

*** Test ...

    ladiff = abs (extr (evtrel - toto5) ORDO) ;
    valtst = maxi ladiff ;

    si(valtst > 1.e-5) ;
       erre 5 ;
    finsi ;

*** Bye ...

    fin ;
```

Test ther-perm **Comments**

1. **Material characteristics**

   evk = evol manu 'T' (prog T1 T2) 'K' (prog K1 K2) ;
   dess evk ;
   macnd = mate mocnd 'K' evk ;

   The conductivity is a function of the temperature and is defined in the evolution (evk). The declaration of the conductivity is done with the operator mate and the values are given in the shape of the evolution object (evk).

2. **Procedure THERMIC**

   tabth = table THERMIQUE ;
   tabth . 'BLOCAGE' = blt ;
   tabth . 'IMPOSE' = ti1 ;
   tabth . 'FLUX' = fl1 ;
   tabth . 'INSTANT(0)' = manu chpo su1 1 T T0 nature diffus ;
   tabth . 'TABCOND' = table ;
   tabth . 'TABCOND' . mocnd = evk ;
   tabth . 'NIVEAU' = 1 ;
   thermic tabth NONLINEAIRE ;

   The procedure THERMIC enables to deal with non linear permanent conditions. It requires some input data which are given in a table of subtype THERMIQUE. The results of the procedure is a field of temperature which is contained in the same table.
   The table tabth has different indexes :
   - 'BLOCAGE' : contains the restraint matrix (RIGIDITE type)
   - 'IMPOSE' : contains the imposed values (CHPOINT type)
   - 'FLUX' : contains the equivalent flow (CHPOINT type)
   - 'INSTANT(0)' : contains the initial temperature field (CHPOINT type)
   - 'TABCOND' : contains the tables of conductivities indexed by the model object of the different subspaces (here tabth . 'TABCOND' . mocnd contains the evolution of the conductivity evk)
   - 'NIVEAU' : indicates the level of message.

3. **Post-treatment**

   tresu = tabth . TEMPERATURE ;
   titr 'Profile of absolute temperature' ;
   evt = evol chpo tresu T li3 ;
   dess evt ;

   The field of temperature generated by the THERMIC procedure is written in tabth . TEMPERATURE (or tresu). It is an object of type CHPOINT. Here the object evt contains the evolution of the component T of the CHPOINT tresu along the line li3.

Chapter 15

THERMO-MECHANICS

15.1 Test thme1 Description sheet

Test name thme1

Calculation type LINEAR THERMAL ANALYSIS AND MECHANIC **Finite element type** QUA8

Topic Thermal diffusion in a bar

The structure is an embedded bar subjected to a thermal diffusion. We study its expansion.

Goal Use of the PASAPAS procedure

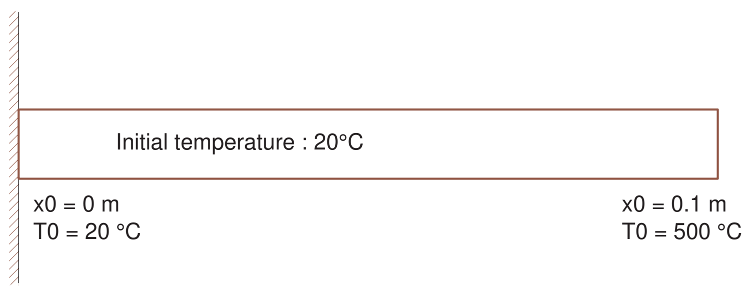
Reference Analytical solution.

Version 97' customer version

Model description

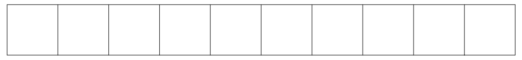
Caracteristic of the bar:

Length of the bar	l	=	0.1
Isotropic conductivity	K	=	14.6
Specific heat	C	=	460
Specific mass	RHO	=	7800
Thermal expansion coefficient	ALPHA	=	$1.\ 10^{-5}$
Young's Modulus	YOUNG	=	$2.\ 10^{11}$
Poisson's Coefficient	NU	=	0.00001

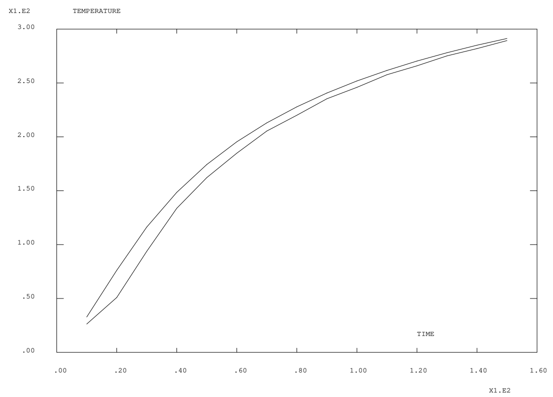


Test thme1 **Results**

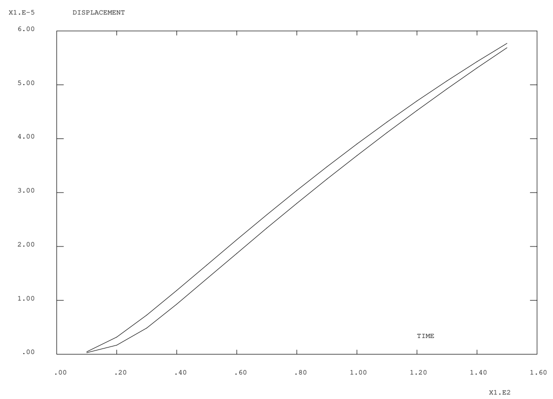
CASTEM FIGURES



THERMO-MECHANIC CALCULATION



EVOLUTION OF THE TEMPERATURE AT THE POINT 8.00000E-02 M



EVOLUTION OF THE DISPLACEMENTAT THE POINT 8.00000E-02 M

```
*          Test Thme1.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
complet = faux;
*
*************************************************
*  pour calcul complet mettre complet à : vrai;
*************************************************
*-----------------------------------------------------
*
* Calcul thermo-mécanique ( mécanique et thermique
* linéaire ). Utilisation de la procédure PASAPAS
*
*
*   |
*   |_____________________________________
*   |                                     |
*   |           T_initial = 20C           | x1 = 0.1 m
*   |_____________________________________| T1 = 500C
*   | x0 = 0 m
*   | T0 = 20C
*     Encastrement
*
*   on étudie la diffusion thermique dans le barreau
*   et sa dilatation
*
*   caracteristiques du barreau :
*
*         l = 0.1m
*         K = 14.6
*         C = 460
*         RHO = 7800
*         ALPHA = 1E-5
*         YOUNG = 2.E11
*         NU = 0.00001
*
*
*   Mailles de 0.01 m de cote
*
*   On compare les résultats à une solution analytique
*
*-----------------------------------------------------

OPTION ECHO 0;
OPTION DIME 2 ELEM QUA8 MODE PLAN DEFO;
*
*-------------------- Creation du maillage ------------
*
dens 1;
PA = 0.   0.  ;
PB = 0.1   0.  ;
PC = 0.1   0.01 ;
PD = 0.   0.01 ;
pp8 = 0.08 0.;
D1 = DROI 10 PA PB ;
D2 = DROI 1  PB PC ;
D3 = DROI 10 PC PD ;
D4 = DROI 1  PD PA ;
L1 = D1 ET D2 ET D3 ET D4;
SU1 = SURF L1 PLANE ;
pp8 = su1 point proc pp8;
*
*-------------------- Définition du modèle ------------
*
MOD1 = MODL SU1 THERMIQUE ISOTROPE CONS UN;
MOD2 = MODL SU1 MECANIQUE ELASTIQUE ISOTROPE CONS DEUX;
MOD = MOD1 ET MOD2 ;
*
*------------------- Définition des matériaux ---------
*
MAT1 = MATR MOD1 K 14.6 'C' 460. RHO 7800.  ;
MAT2 = MATR MOD2 ALPHA 1E-5 YOUN 2E+11 NU 0.00001;
MAT = MAT1 ET MAT2 ;
*
*------------------- blocages mécaniques --------------
*
CL1 = BLOQ UY SU1 ;
CL2 = BLOQ UX D4 ;
CL_ME = CL1 ET CL2;
*
*------------------- blocages thermiques --------------
*
CL3 = BLOQ T D2;
CL4 = BLOQ T D4;
CL_TH = CL3 ET CL4;
*
*------------------- températures imposées ------------
*
TEM1 = DEPI CL3 500.;
TEM2 = DEPI CL4 20.;
EV1 = EVOL MANU x (PROG 0. 200.) y (prog 1. 1.);
CHA = CHAR TIMP (TEM1 + TEM2) EV1;
*
*------------------- température initiale -------------
*
TEM0 = MANU CHPO SU1 1 T 20.;
*
*--------------- creation de la table pour pasapas ----
*
ta = table;
ta.modele = mod;
ta.caracteristiques = mat;
ta.blocages_mecaniques = cl_me;
ta.blocages_thermiques = cl_th;
ta.temperatures = table;
ta.temperatures . 0 = tem0;
ta.chargement = cha;
ta.procedure_thermique = LINEAIRE;
*ta.temps_calcules = prog 0. pas 5. 150.;
ta.temps_calcules = prog 0. pas 10. 150.;
*
pasapas ta;
*
*----------------------- Depouillement ----------------
*
a = (14.6 / 7800.)/460.;
x = 0.08;
er1 = 0.;
er2 = 0.;
nb = 0;
tpr1 = prog;
tpr2 = prog;
tpr3 = prog;
tep_th = prog;
dep_th = prog;
idim = (dime (ta.temps)) - 1;
solteenr = prog
   32.779      76.142      116.26      148.47
   174.30      195.40      212.98      227.88
   240.70      251.85      261.65      270.33
   278.08      285.03      291.30  ;
 soldeenr = prog
 4.51064E-07  3.18778E-06  7.26909E-06  1.18402E-05
 1.65525E-05  2.12583E-05  2.58885E-05  3.04069E-05
 3.47909E-05  3.90251E-05  4.30987E-05  4.70048E-05
 5.07398E-05  5.43030E-05  5.76962E-05;
```

```
repeter bou idim;
   ite = &bou;
   te = ta.temps.ite;
   dep = ta.deplacements.ite;
   tem = ta.temperatures.ite;
   tep1 = extr tem T pp8;
   dep1 = extr dep UX pp8;
   tpr1 = tpr1 et (prog tep1);
   tpr2 = tpr2 et (prog dep1);
   tpr3 = tpr3 et (prog te);
si complet;
   co1 = erf((0.1 - x)/(2 * (( a * te )**0.5)));
   co2 = erf(0.1/(2.*(( a * te )**0.5)));
   sol_te = 500. - ((500. - 20.)*co1/co2);
sinon;
    sol_te = extr solteenr ite;
finsi;
   tep_th = tep_th et (prog sol_te);
si complet;
   mi = prog;
   mi1 = prog ;
   repeter bou2  21;
      ite2 = &bou2 - 1;
      te1 = x * ite2 / 20;
      mi = mi et (prog te1);
      y1 = erf((0.1 - te1)/(2.*(( a * te )**0.5)));
      mi1 = mi1 et (prog y1);
   fin bou2;
   ev5 = evol manu xx mi y mi1;
   res = extr (somm ev5) 1;
   sol_de = 1E-5*(500. - 20.)/co2*((x * co2) - res);
sinon;
    sol_de = extr soldeenr ite;
finsi;
   dep_th = dep_th et (prog sol_de);
   si ((flot ite) >  ((flot idim)/2) );
     er1 = er1 + (abs ((sol_te - tep1)/sol_te) * 100.);
     er2 = er2 + (abs ((sol_de - dep1)/sol_de) * 100.);
      nb = nb + 1;
   finsi;
fin bou;
*
er1 = er1/nb;
er2 = er2/nb;
mess 'erreur sur la température :'er1'%';
mess 'erreur sur le deplacement :'er2'%';
*
*lvar =text 'EVOLUTION DE LA TEMPERATURE AU POINT'x'm';
*titre lvar;
*ev_te = evol manu temps tpr3 temperature tpr1;
*ev_tet = evol manu temps tpr3 temperature tep_th;
*dess (ev_te et ev_tet);
*lvar = text 'EVOLUTION DU DEPLACEMENT AU POINT'x'm';
*titre lvar;
*ev_de = evol manu temps tpr3 deplacement tpr2;
*ev_det = evol manu temps tpr3 deplacement dep_th;
*dess (ev_de et ev_det);
*
si ( er1 > 2. ) ;
  mess 'Résultat incorrect' ;
  erreur 5 ;
finsi ;
si ( er2 > 5. ) ;
  mess 'Résultat incorrect' ;
  erreur 5 ;
finsi ;
*
fin;
```

Test thme1 **Comments**

- The THERMIQUE model

```
MOD1 = MODE SU1 THERMIQUE ISOTROPE CONS UN ;
MAT1 = MATE MOD1 K 14.6 'C' 460. RHO 7800.  ;
```

The thermic formulation works in three cases: 'ISOTROPE', 'ORTHOTROPE' and 'ANISOTROPE'. Here we study only an isotropic material.

Parameter names for a THERMIQUE ISOTROPE formulation

'K'	isotropic conductivity
'RHO'	specific mass
'C'	specific heat
'H'	exchange coefficient

Parameter names for a THERMIQUE ORTHOTROPE formulation

- Shells (COQ2, COQ3, COQ4, COQ6, COQ8)

'K1','K2','K3'	thermal conductivities
'RHO'	density
'C '	specific heat
'H '	exchange coefficient

- Three-dimensional solid elements

'K1','K2','K3'	thermal conductivities
'RHO'	density
'C '	specific heat
'H '	exchange coefficient

- Two-dimensional solid elements

two-dimensional and axisymmetrical	Fourier's series
'K1','K2' 'RHO','H','C'	'K1','K2','K3' 'RHO','H','C'

Parameter names for a THERMIQUE ANISOTROPE formulation

- Three-dimensional solid elements

'K11','K21','K22' 'K31','K32','K33'	terms independent of the anisotropic conductivities matrix
'RHO'	density
'H'	exchange coefficient
'C'	specific heat

- Two-dimensional solid elements

two-dimensional and axisymmetrical	Fourier's series
'K11','K21','K22' 'RHO','H','C'	'K11','K21','K22' 'K33' 'RHO','H','C'

Note: The geometrical properties must be input simultaneously with the material properties when carrying out a thermal analysis shells or a mechanical analysis thick orthotropic shells.

15.2 Test thpl1 Description sheet

Test name thpl1

Calculation type KINEMATIC ELASTOPLASTIC MATERIAL

Finite element type QUA4

Topic A cylindrical bar subjected to a field of temperature The structure is a 100-mm long and 20-mm

diameter cylindrical bar. It is axially embedded at the two tips and subjected to a constant field of temperature (500C).
We solve the problem in axisymetrical mode. We use the PASAPAS non-linear procedure because of the plastic material.

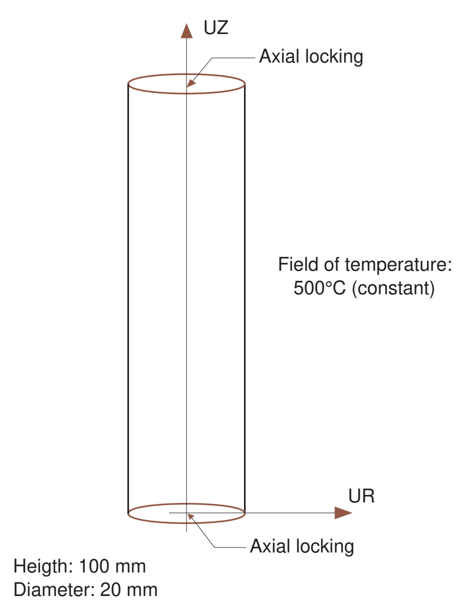
Goal Calculate the axial stress and the axial plastic deformation.

Reference Test de la Commission V.P.C.S. Groupe Thermo-plasticite et Statique non-lineaire
Test numero SSNA01/90

Version 97' customer version

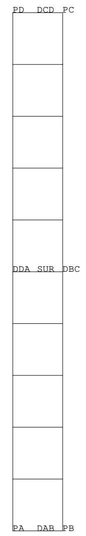
Model description

Young's modulus	E	=	$200\ 10^3$
Poisson's coefficient	NU	=	0.3
Elastic limit	SYGY	=	300
Stress hardening modulus	H	=	2000
Thermal expansion coefficient	ALPHA	=	$1.2\ 10^{-5}$



Test thpl1 **Results**

CASTEM FIGURES



THPL1 : MESH

THPL1 : DEFORMED SHAPE

```
*          Test Thpl1.dgibi: Jeux de données        *
*          ---------------------------------        *
*                                                   *
* CAS TEST DU 92/03/20    PROVENANCE : TC1
SAUT PAGE;
*
*****************************************************
*                                                   *
*    TEST THPL1                                     *
*                                                   *
*    BARREAU CYLINDRIQUE SOUMIS A UN CHAMPS         *
*              DE TEMPERATURE                       *
*                                                   *
*    Test de la Commission V.P.C.S.                 *
*    Groupe Thermo-plasticite  Statique non-lineaire *
*    Test numero SSNA01/90                          *
*                                                   *
*        Un barreau cylindrique de 100 mm de hauteur *
*    et de 20 mm de diametre est bloque axialement  *
*    a ces deux extremites. Il est soumis a un champ *
*    constant de temperature, de 500  degres celsius. *
*                                                   *
*        On traite le probleme en mode axisymetrique. *
*    On utilise la procedure PASAPAS car le materiau *
*    est plastique.                                 *
*                                                   *
*        On se propose de calculer                  *
*               - d'une part la contrainte axiale   *
*            et - d'autre part la deformation       *
*                 plastique axiale.                 *
*                                                   *
*****************************************************
*
OPTI ECHO 0;
*
OPTION DIME 2;
OPTION MODE AXIS ELEM QUA4;
*
GRAPH = 'N';
TEMPS;
*
*===================================================*
*                 geometrie : maillage              *
*===================================================*
*
* Dimensions en millimetres.
*
RA = 0.0 ;  ZA = 0. ;     PA = RA ZA ;
RB = 10. ;  ZB = 0. ;     PB = RB ZB ;
RC = 10. ;  ZC = 100.0 ;  PC = RC ZC ;
RD = 0.0 ;  ZD = 100.0 ;  PD = RD ZD ;
*
DAB = DROI 1  PA PB;
DBC = DROI 10 PB PC;
DCD = DROI 1  PC PD;
DDA = DROI 10 PD PA;
*
SUR = DALL DAB DBC DCD DDA PLAN;
*
ELIM SUR 0.001;
*
SI (NEG GRAPH 'N');
   TITR 'THPL1 : MAILLAGE';
   TRAC QUAL SUR;
FINSI;
*
ELSUR = NBEL SUR;
NOSUR = NBNO SUR;
```

```
*
*===================================================*
*                conditions aux limites             *
*===================================================*
*
CL1 = BLOQUE UZ DAB ;
CL2 = BLOQUE UZ DCD ;
CL = CL1 ET CL2 ;
*
*===================================================*
*                 modl - materiau                   *
*===================================================*
*
MOD = MODL SUR MECANIQUE ELASTIQUE PLASTIQUE
      CINEMATIQUE QUA4;

MAT = MATR MOD YOUN 200E3 NU 0.3 SIGY 300 H 2000
      ALPH 1.2E-5;
*
*===================================================*
*          champs relatifs a la procedure PASAPAS   *
*===================================================*
*
* Liste des pseudo-temps ou l'on desire des resultats
*
T0 = 0. ; T1 = 1. ;
LISTE = PROG T0 T1 ;
*
* Champs-point de temperature aux instants T0 et T1.
*
TEMP = TABLE;TEMPER = TABLE;
TEMP . 0 = T0;TEMP . 1 = T1;
TEMPER . 0 = MANU CHPO SUR 1 T 0.;
TEMPER . 1 =  MANU CHPO SUR 1 T 500.;
CHA2 = CHAR 'T' TEMP TEMPER;
*
* Chargement mecanique de la structure: chargement nul.
*
LFORCE = PROG 0. 0.;
FOR = FORCE FR 1000. PC;
EVOL1 = EVOL MANU TEMPS LISTE FORCE LFORCE;
CHA1 = CHAR 'MECA' FOR EVOL1;
*
* Initialisation de la table TAB1 :
* parametres pour la procedure PASAPAS
*

TAB1 = TABLE;
TAB1.'CHARGEMENT' = (CHA1 ET CHA2);
TAB1.'MODELE' = MOD;
TAB1.'CARACTERISTIQUES' = MAT;
TAB1.'BLOCAGES_MECANIQUES' = CL;
TAB1.'TEMPS_CALCULES' = LISTE;
TAB1.'TEMPS_SAUVES' = LISTE;

*
* procedure PASAPAS
*
PASAPAS TAB1;
*
*===================================================*
*            extractions des resultats              *
*===================================================*
*
* Contraintes
*
nn = 1;
CHEL1 = TAB1. 'CONTRAINTES'.nn;
```

```
*

CHAM2 = CHAN NOEUD MOD CHEL1;
CHPO1 = CHAN CHPO MOD CHAM2;
*
* Extraction du la contrainte SMZZ
*
SMZZ1 = EXTR CHPO1 SMZZ PA;
SMZZ2 = -309.0 ;
*
* Deformations plastiques
*
CHEL10 = TAB1. 'DEFORMATIONS_INELASTIQUES' .nn;
*
* Interpolations aux noeuds du maillage
* Utilisation des nouveaux chamelems.
*

CHAM20 = CHAN NOEUD MOD CHEL10;
CHPO10 = CHAN CHPO MOD CHAM20;
*
* Extraction de la deformation plastique EPZZ
*
EPZZ1 = EXTR CHPO10 EIZZ PA;
EPZZ2 = -.4455E-2 ;
*
*====================================================*
*               affichage des resultats              *
*====================================================*
*
SAUT 1 LIGN ;
MESS ' RESULTATS ';
MESS ' --------- ';
SAUT 1 LIGN ;
*
MESS '   1) Contraintes ' ;
MESS '      ----------- ' ;
SAUT 1 LIGN ;
*
MESS '      Contrainte theorique smzz : ' SMZZ2 'Mpa' ;
MESS '      Contrainte calculee smzz  : ' SMZZ1 'Mpa' ;
ERG1 = ( ABS ((SMZZ2 - SMZZ1) / SMZZ2 )) * 100. ;
MESS '              Soit un ecart de  :' ERG1 '%';
SAUT 1 LIGN;
MESS '   2) Deformations plastiques ';
MESS '      ---------------------- ' ;
SAUT 1 LIGN ;
*
MESS '       Deformation plastique theorique : ' EPZZ2;
MESS '       Deformation plastique calculee  : ' EPZZ1;
ERG2 = ( ABS ((EPZZ2 - EPZZ1) / EPZZ2 )) * 100. ;
MESS
'                       Soit un ecart de :' ERG2 '%';
*
SAUT 1 LIGN;
MESS '   Maillage ';
MESS '   -------- ';
MESS '     Nombre de noeuds        : ' NOSUR;
MESS '     Nombre d elements (qua4): ' ELSUR;
SAUT 1 LIGN;
*
SI (NEG GRAPH 'N');
  DEP1 = TAB1. 'DEPLACEMENTS' . nn;
  DEF0 = DEFO SUR DEP1 0.;
  DEF1 = DEFO SUR DEP1 10. ROUGE;
  TITR 'THPL1 : DEFORMEE';
  TRAC (DEF0 ET DEF1);
FINSI;
*
*====================================================*
*              code fonctionnement et fin            *
*====================================================*
*
* Maximum des erreurs relatives
*
ERGMAX = MAXI (PROG ERG1 ERG2);
*
* Il y a erreur si un des pourcentage erg*
* est superieur à 5%.
*
SI (ERGMAX <EG 5.);
     ERRE  0 ;
SINON;
     ERRE  5 ;
FINSI;
*
SAUT 1 LIGN;
TEMPS ;
SAUT 1 LIGN;
*
FIN;
*
```

Test thpl1 **Comments**

- The plastic model with CINEMATIQUE LINEAIRE stress hardening

```
MOD = MODE SUR MECANIQUE ELASTIQUE PLASTIQUE
      CINEMATIQUE QUA4 ;
MAT = MATE MOD YOUN 200E3 NU 0.3 SIGY 300 H 2000
      ALPH 1.2E-5 ;
```

This non-linear behavior works with the Von Mises model with linear kinematic stress hardening. The parameters to be input are the elastic limit (SIGY) and the stress hardening modulus (H).

Chapter 16

MAGNETO-STATIC

16.1 Test c2d93 Description sheet

Test name c2d93

Calculation type 2D AXISYMMETRIC VECTOR POTENTIAL Computation of magnetic field with non linear material **Finite element type** TRI6 QUA8

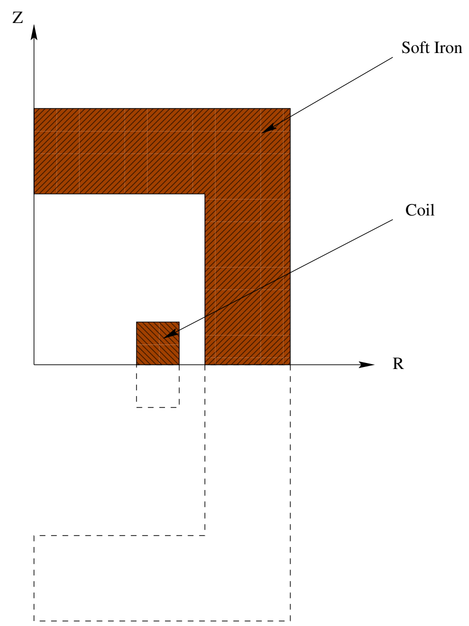
Topic

A circular coil of rectangular cross section is embeded in an iron box. **Goal** Calculate vector potential and field in the cavity.

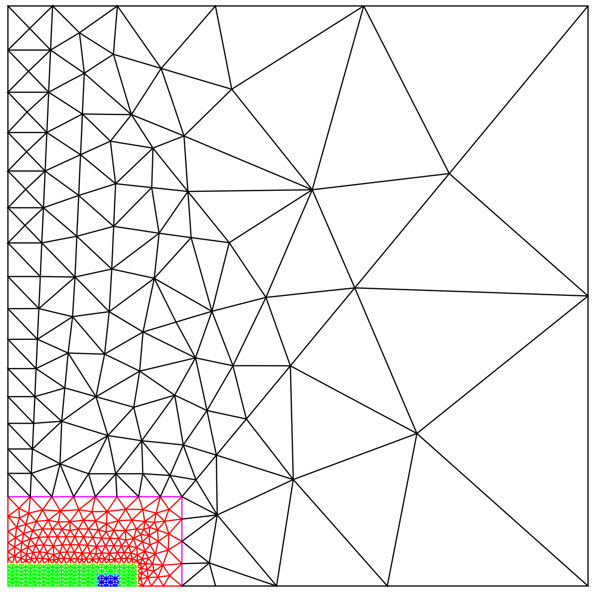
Reference Analytical solution.

Version 97' customer version

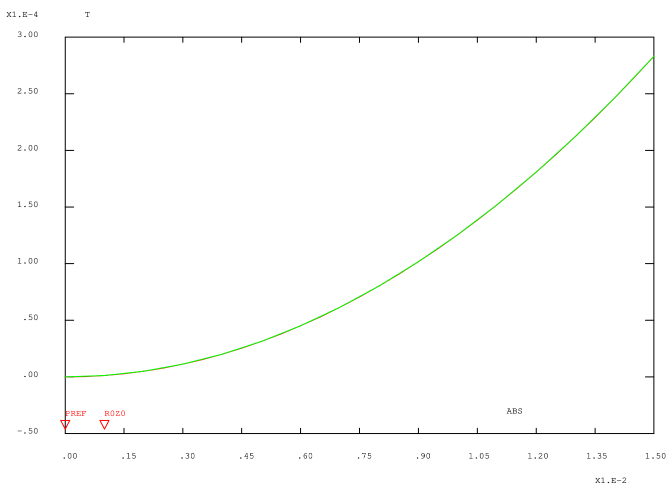
Model description Soft iron curve for B(H).



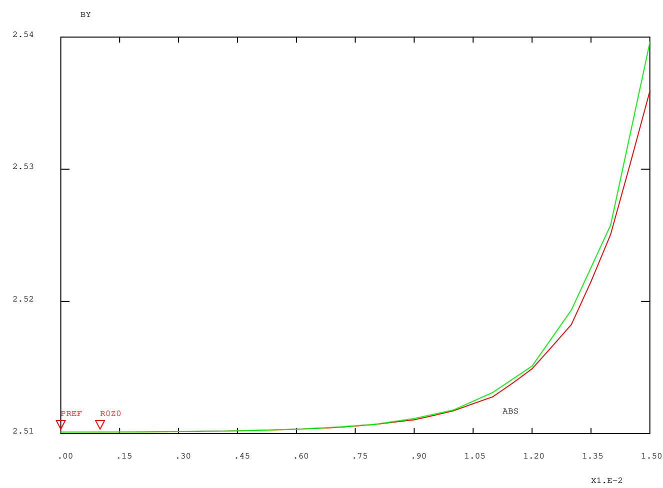
CASTEM FIGURES



MESH 2D



POTENTIAL BEFORE AND AFTER SMOTHING



BY COMPONENT BEFORE AND AFTER SMOTHING

```
***********************************************************************
*
*       2D   AXISYMMETRIC  MAGNETIC FIELD COMPUTATION
*
*       Formulation  : VECTOR POTENTIAL
*
*                      NON LINEAR MATERIAL
*
***********************************************************************
 OPTION DIME 2  ELEM  TRI6  COUL VERT echo 1 ;
 GRAPH ='N' ;
* ------------------------Mesh ---------------------------
 R1 =20. ; R2= 25. ; R3= 27. ; R4= 29. ; R5=39.;R6 = 130.;
 Z1= 2.5;  Z2=  5. ; Z3= 20. ; Z4= 130.;Z31=30.;Z32=50.;
 DI1= 1.; DI2=5.;DI3=60. ;DI4= 10.;NET2= 6;

  DENS DI1 ;
 OZ3= 0.  Z3; OZ2= 0.  Z2;OZ1= 0.  Z1;OZ4=0. Z4;
 OO= 0.  0.;R0Z0= DI1  0 ;
 R1Z0=  R1  0. ;R2Z0= R2  0. ;R3Z0 = R3  0. ;R4Z0 = R4  0.;
 R0Z2= DI1 Z2 ; R4Z2 = R4 Z2 ;
  DENS DI2 ;
 R5Z0=  R5  0. ;R5Z3= R5 Z3 ;R0Z3 =DI2 Z3 ;OZ3 = 0. Z3 ;
  DENS DI3 ;
 R6Z0=R6  0.; R6Z4=R6 Z4 ;
  DENS 10 ;
 R2Z3= R2  Z3 ; R1Z3= R1  Z3; R4Z1 = R4   Z1;
 R3Z1= R3  Z1 ;
 DENSITE DI4 ;
 R0Z4 = DI4 Z4 ; OZ4 = 0. Z4 ;
 NTRA= 1;
 FLAN1 =( D OO R0Z0 D R1Z0 ) TRAN DINI DI1 DFIN DI1 ( OZ1 MOINS OO )
             TRAN DINI DI1 DFIN DI1 ( OZ2 MOINS OZ1) COUL VERT ;
*-------------------------  COIL SURFACE -------------------
 BOBI  =( D R1Z0 R2Z0 ) TRAN DINI DI1 DFIN DI1 ( OZ1 MOINS OO)
                                                        COUL BLEU ;
*-----------------------------------------------------------
 FLAN2 =(INVE( BOBI COTE 3)) TRAN DINI DI1 DFIN DI1
                                 ( OZ2 MOINS OZ1) COUL VERT ;
 FLAN3 =( D R2Z0 R3Z0 ) TRAN DINI DI1 DFIN DI1 ( OZ1 MOINS OO)
             TRAN DINI DI1 DFIN DI1 ( OZ2 MOINS OZ1) COUL VERT ;
 FLAN4 =( D R3Z0 R4Z0 ) TRAN DINI DI1 DFIN DI1 ( OZ1 MOINS OO)
             TRAN DINI DI1 DFIN DI1 ( OZ2 MOINS OZ1) COUL VERT ;
*---------------------IRON ----------------------------------
 FER = (D OZ2 R0Z2 D R4Z2 D 6 R4Z0 D R5Z0 D R5Z3 D R0Z3 D OZ3 D OZ2 )
 SURF PLANE COUL ROUG ;
*---------------------EXTERNAL AIR BOX ------------------
 FLAN5 = ( D   R5Z0 R5Z3  D R6Z4 D R6Z0 D R5Z0 ) SURF PLANE ;
*
 FLAN6 =( D R0Z3  R5Z3 D R6Z4  D R0Z4 D R0Z3 ) SURF PLANE
                         COUL VERT ;
 TUB6 = (D OZ3 R0Z3 D R0Z4 D OZ4 D OZ3) SURF PLANE COUL vert ;
 AIREXT = (FLAN6 ET TUB6 ET FLAN5 ) coul blan ;
 AIRIN = ((FLAN1 ET FLAN2 ET FLAN3  ET FLAN4 ) coul vert) et BOBI ;
 TOUT = AIRIN ET AIREXT ET FER ; ELIM .2 TOUT ;
*
*---------------  BOUNDARIES ---------------------------------
*
 ENPP = CONTOUR TOUT  ;
 CEXTR=ENPP POINTS DROITE   (130.  0.) (130. 100.)  .1 ;
 CEXTH=ENPP POINTS DROITE   ( 0. 130.) (10.  130.)  .1 ;
 AXE=ENPP POINTS DROITE   ( 0.  0. ) ( 0. 100. )  .1  ;
 AX1 = (AIRIN ET FER) POINT DROITE ( 0.  0. ) ( 0. 100. ) .1 ;
 AX2 = DIFF ( CHAN POI1 AXE) (CHAN POI1 AX1) ;
*
*-----------------------------------------------------------------------
*        FORMER DESCRIPTION IN  MILLIMETRES
```

```
*---------------- SHIFT FOR METERS -----------------------------------
*
 deplacer tout homo  .001  (0. 0.) ;
*
 SI (NEG GRAPH  N ) ;
 TITRE  ' MESH 2D ' ;
 TRAC  tout  ;
 FINSI ;
*----------------------------------------------------------------------
 MU0= 4. * PI * 1.E-7 ;
*
*------------------CURENT DESCRIPTION ---------------------------------
 TABCOUR= TABLE ;
 DESCOUR TABCOUR 1 BOBI 'AMP' 800.E6 ;
 TABB = TABLE ;
*----------------- AXISYMMETRIC  PROBLEM ------------------------------
 TABB.'AXI'= VRAI  ;
  NF = FER NBEL ;
   FER1 = FER ELEM ( LECT 1 PAS 1 30 )  ;
   FER2 = FER ELEM ( LECT 31 PAS 1 NF ) ;
*------------------MATERIALS  -----------------------------------------
 KEVOL = H_B MU0    ;
 TABMAT = TABLE ;
    OBFER1 = FER1 MODE THERMIQUE ISOTROPE ;
    OBFER2 = FER2 MODE THERMIQUE ISOTROPE ;
   STN = TABLE ; STN.EV1 = KEVOL   ;
   TABMAT.OBFER1 = STN ;
   STN = TABLE ; STN.EV1 = KEVOL   ;
   TABMAT.OBFER2 = STN ;
 TABB.'TABNUSEC' = TABMAT ;
 TABB.'MUAIR' =MU0 ;
*NON LINEAR MATERIAL CAN BE TRAITED AS A SUPER ELEMENT OR NOT
*
  isuper= 1 ;
 SI ('EGA' isuper 1 ) ;
 TABB.'AIRSUP' = AIREXT ;
 TABB.'MAITRES' = ( FER CONTOUR) COMPRIS R5Z0 OZ3 ;
 TABB.'ENCS' = CEXTR ET CEXTH ET AX2 ;
 TABB.'BLOQUE' = BLOQUER 'T' AX1 ;
 TABB.'AIR' = AIRIN ;
 SINON ;
 TABB.'AIR' = (AIRIN ET AIREXT) ;
 TABB.'BLOQUE' = BLOQUER 'T' (CEXTR ET CEXTH  ET AXE ) ;
 'FINSI' ;
 TABB.'COUR' = TABCOUR ;
*
* ------POTENTIAL  COMPUTATION OR FIRST STEP IF NON LINEAR PROBLEM ---
*----------
 POT_VECT  TABB 'SOLIN' ;
*-----
*-------------------------------------------------------------------
 SOL1 = (TABB.'POTENTIEL' ) ENLEVER LX ;
 RAY = FLAN1 COTE 1 ;
 RAY2 = D 10 (0. .0001) ( .020  .0001) ;
*------------SOME  POST TRAITMENT   B COMPUTATION ----------------
 BB = INDUCTIO  AIRIN  SOL1  VRAI ;
 PREF = AIRIN POINT PROCHE ( 0. 0. ) ;
 BY0 = EXTR BB 'BY'  PREF  ;
 BY10 = EXTR BB 'BY' ( AIRIN POINT PROCHE ( .010 0.)) ;
 TITRE ' COMPOSANTE BY AVANT ET APRES LISSAGE ' ;
 EVB1  = EVOL ROUG  CHPO  BB    'BY'  RAY  ;
*----------   polynomial  smothing  if  wanted  --------------------
 CHLIS = PROI  POLY  TOUT RAY2   SOL1 1 AXIS ;
 BBY = (EXCO CHLIS 'BY' ) NOMC  'BY' ;
 BY01 = EXTR BBY 'BY' (RAY2 POINT INITIAL) ;
 BAT1  = 800.e6  * 5.e-3 * 5.e-3  * MU0 /10.e-3 ;

 option echo 0 ;
```

```
 MESS '*************************************************************
 MESS '*  CIRCULAR COIL  internal radius 20 mm '
 MESS '*        total    cross section  5*5 mm '
 MESS '*         SYMMETRY BY HORIZONTAL PLANE  '
 MESS '*  AMPERE  mufer >> muo  all AMPERE*TURNS in the GAP of 10 mm
 MESS '*
 MESS '*       J * EP * HAUT = B/ MU0 * e
 MESS '*   WAITED :   By = '  bat1 '  TESLAS
 MESS ' ************************************************************
 MESS '  NUMERICAL DERIVATION    BY AU CENTRE ' BY0   ;
 MESS '  SMOTHED SOLUTION        BY AU CENTRE ' BY01   ; *
 MESS ' ****************************************';
********************************************************************
   RAP = ABS ((BY01 - bat1 ) / bat1  );
       SI ( RAP  > .01 ) ; ERREUR 5 ; FINSI ;
********************************************************************
 SI (NEG  GRAPH  N ) ;
 TITRE ' POTENTIAL   BEFORE AND AFTER   SMOTHING ' ;
 EVPO1 = EVOL ROUG  CHPO  SOL1   'T'   RAY  ;
 EVPO2 = EVOL VERT  CHPO  CHLIS    'A' RAY2  ;
 dess (evpo1 et evpo2) xbor 0. (R1 * 0.001 * .75);
 TITRE ' BY COMPONENT  BEFORE AND AFTER SMOTHING  ' ;
 EVB1  = EVOL ROUG  CHPO  BB    'BY'   RAY  ;
 EVB2  = EVOL VERT  CHPO  CHLIS  'BY'  RAY2  ;

 dess (evb1 et evb2) xbor 0. (R1 * 0.001 * .75) ;
  FINSI ;
*
**-----------------    NON LINEAR  COMPUTATION  -------------------   ;
*      filling
 TABB.SOUSTYPE='THERMIQUE' ;
 TABB.CRITERE =1.E-4 ;
 TABB.NITER   =1;
 TABB.'OME' = .1 ;
*-----------  test shortened to 2 iterations ----------------------
 TABB.ITERMAX=2;
 TABB.NIVEAU =1;
*
*-------------------------------
  MAG_NLIN  TABB  ;
*-------------------------------
*
 SOL2 = enlever (TABB.'POTENTIEL' ) LX ;
 BB = INDUCTIO  AIRIN SOL2  VRAI ;
 BY02 = EXTR BB 'BY'  PREF ;
 CHLIS = PROI  POLY  TOUT RAY2   SOL2 1 AXIS ;
 BBY = (EXCO CHLIS 'BY' ) NOMC  'BY' ;
 BY03 = EXTR BBY 'BY' (RAY2 POINT INITIAL) ;
 BAT = 2.4106 ;
 MESS ' ****************************************';
 MESS '  EXPECTED SOLUTIONS   AT CENTER   BY  ' BAT      ;
 MESS '  NUMERICAL DERIVATION             BY  ' BY02     ;
 MESS '  SMOTHED SOLUTION                 BY  ' BY03     ;
 MESS ' ****************************************';
    itest = 1 ;
*--------------------GOOD WORKING MESSAGE -----------------------
    SI (EGA ITEST 1 ) ;
*------- test   shortened ---------------------------------------
   RAP = ABS ((BY03 - BAT ) / BAT);
       SI ( RAP  > .01 ) ; ERREUR 5 ; FINSI ;
*---------------------------------------------------------------
   SINON ;
*------  COMPUTATION TILL CONVERGENCE OR UP TO 100 MORE ITERATION
 TABB.ITERMAX=100;
  MAG_NLIN  TABB  ;
*---------------------------------------------------------------
 SOL2 = enlever (TABB.'POTENTIEL' ) LX ;
 BB = INDUCTIO  AIRIN SOL2  VRAI ;
```

```
BY02 = EXTR BB 'BY'  PREF ;
CHLIS = PROI POLY  TOUT RAY2   SOL2 1 AXIS ;
BBY = (EXCO CHLIS 'BY' ) NOMC  'BY' ;
BY03 = EXTR BBY 'BY' (RAY2 POINT INITIAL) ;
BAT = 2.3468 ;
MESS ' ****************************************';
MESS '  EXPECTED SOLUTION ON AXIS      BY   ' BAT      ;
MESS '  NUMERICAL DERIVATION  BY AU CENTRE ' BY02     ;
MESS '  SMOTHED SOLUTION      BY AU CENTRE ' BY03     ;
MESS ' ****************************************';
*******************************************************************
  RAP = ABS ((BY03 - BAT ) / BAT );
      SI ( RAP  > .01 ) ; ERREUR 5 ; FINSI ;
*******************************************************************
 SI (NEG GRAPH N )  ;
TITRE ' POTENTIAL   BEFORE AND AFTER   SMOTHING ' ;
EVPO1 = EVOL ROUG  CHPO  SOL2   'T'   RAY  ;
EVPO2 = EVOL VERT  CHPO  CHLIS    'A' RAY2  ;
dess (evpo1 et evpo2) xbor 0. (R1 * 0.001 * .75);
TITRE ' BY COMPONENT  BEFORE AND AFTER SMOTHING ' ;
EVB1  = EVOL ROUG  CHPO  BB    'BY'  RAY  ;
EVB2  = EVOL VERT  CHPO  CHLIS  'BY'  RAY2  ;
dess (evb1 et evb2) xbor 0. (R1 * 0.001 * .75);
FINSI ;
  FINSI ;
FIN ;
```

16.2 Test c3d93 Description sheet

Test name c3d93

Calculation type 3D SCALAR POTENTIAL COMPUTATION OF MAGNETIC FIELD Non Linear Material. 2 Scalar potentials method (see Trowbridge - Simkin)

Finite element type CUB8 or CU20

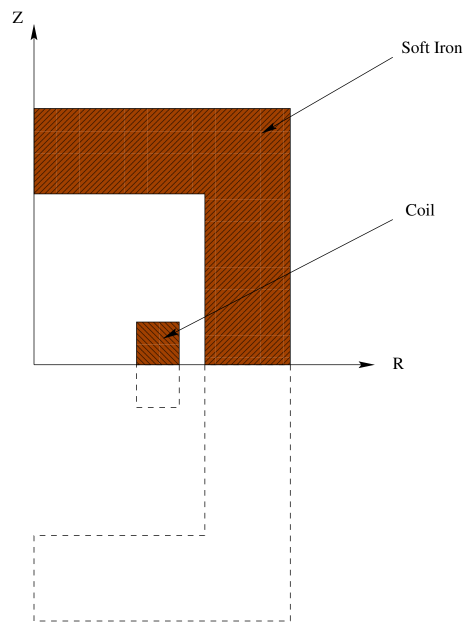
Topic A circular coil of rectangular cross section is embeded in an iron box.

Goal Calculate scalar potential and field in the cavity.

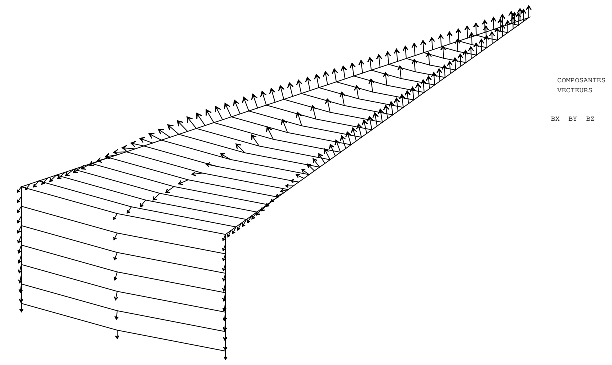
Reference Analytical solution.

Version 97' customer version

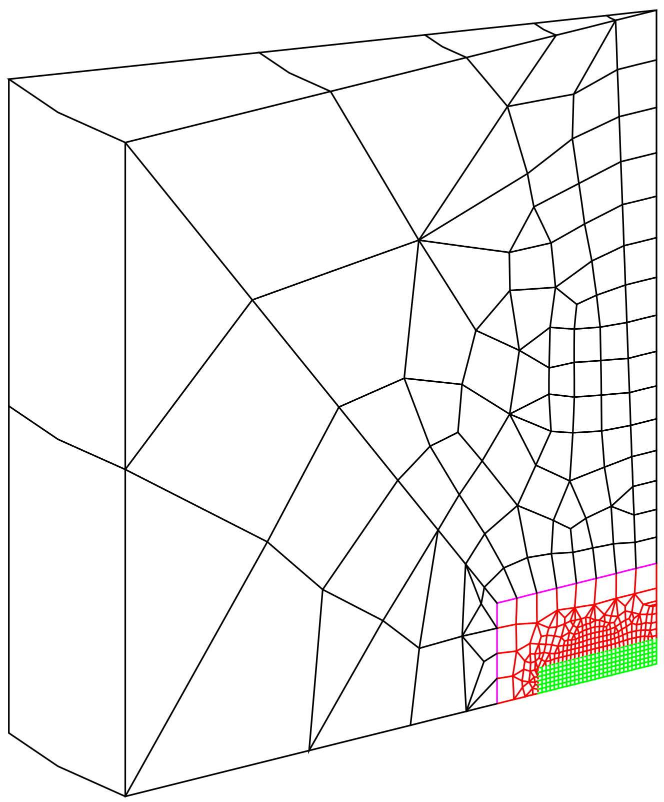
Model description Soft iron curve for B(H).



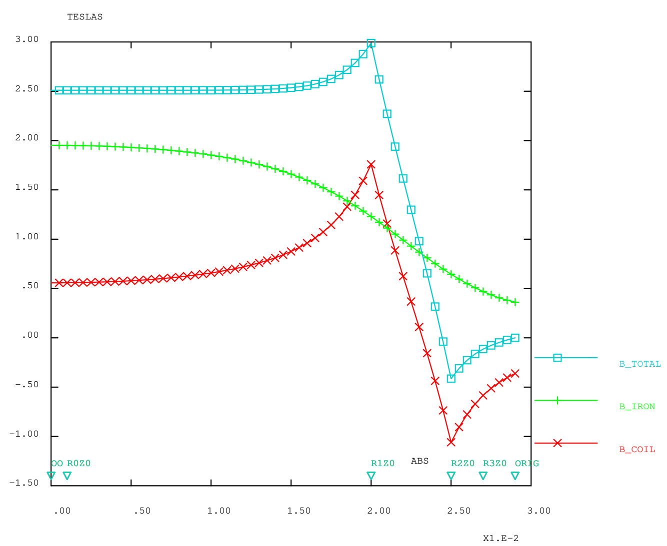
CASTEM FIGURES



HS ON SEPARATION SURFACE



MESH 3D



BZ(R) for Z=0

```
*******************************************************************
*       3D MAGNETIC FIELD COMPUTATION  2 POTENTIALS METHOD
*
*         REDUCED POTENTIAL  (VOLUME CONTAINING INDUCTORS)
*         TOTAL POTENTIAL    (VOLUME WHITHOUT  CURRENTS  )
*
*******************************************************************
option echo 1 ;
    TYPEL = 2 ;
GRAPH = N ;
   si ( ega typel 2 ) ;
 OPTION DIME 2  ELEM  QUA8  COUL VERT echo 0 ;
   sinon ;
 OPTION DIME 2  ELEM  QUA4  COUL VERT echo 0 ;
   finsi ;
*---------------------------MESH ---------------------------
 R1 =20. ; R2= 25. ; R3= 27. ; R4= 29. ; R5=39.;R6 = 130.;
 Z1= 2.5;  Z2=  5. ; Z3= 20. ; Z4= 130.;Z31=30.;Z32=50.;
 DI1= 1.; DI2=5.;DI3=60. ;DI4= 10.;NET2= 6;

  DENS DI1 ;
 OZ3= 0.  Z3; OZ2= 0.  Z2;OZ1= 0.  Z1;OZ4=0. Z4;
 OO= 0.  0.;ROZO= DI1  0 ;
 R1Z0=  R1  0. ;R2Z0= R2  0. ;R3Z0 = R3  0. ;R4Z0 = R4  0. ;
 ROZ2= DI1 Z2 ; R4Z2 = R4 Z2 ;
  DENS DI2 ;
 R5Z0=  R5  0. ;R5Z3= R5 Z3 ;ROZ3 =DI2 Z3 ;OZ3 = 0. Z3 ;
  DENS DI3 ;
 R6Z0=R6  0.; R6Z4=R6 Z4 ;
  DENS 10 ;
 R2Z3= R2  Z3 ; R1Z3= R1  Z3; R4Z1 = R4   Z1;
 R3Z1= R3  Z1 ;
 DENSITE DI4 ;
 ROZ4 = DI4 Z4 ; OZ4 = 0. Z4 ;
 NTRA= 1;
 FLAN1 =( D OO ROZO D R1Z0 ) TRAN DINI DI1 DFIN DI1 ( OZ1 MOINS OO )
            TRAN DINI DI1 DFIN DI1 ( OZ2 MOINS OZ1) COUL VERT ;
 BOBI  =( D R1Z0 R2Z0 ) TRAN DINI DI1 DFIN DI1 ( OZ1 MOINS OO)
                                                     COUL BLEU ;
 FLAN2 =(INVE( BOBI COTE 3)) TRAN DINI DI1 DFIN DI1
                             ( OZ2 MOINS OZ1) COUL VERT ;
 FLAN3 =( D R2Z0 R3Z0 ) TRAN DINI DI1 DFIN DI1 ( OZ1 MOINS OO )
            TRAN DINI DI1 DFIN DI1 ( OZ2 MOINS OZ1) COUL VERT ;
 FLAN4 =( D R3Z0 R4Z0 ) TRAN DINI DI1 DFIN DI1 ( OZ1 MOINS OO )
            TRAN DINI DI1 DFIN DI1 ( OZ2 MOINS OZ1) COUL VERT ;
  FER = (D OZ2 ROZ2 D R4Z2 D 6 R4Z0 D R5Z0 D R5Z3 D ROZ3 D OZ2 )
  SURF PLANE COUL ROUG ;

 FLAN5 = ( D   R5Z0 R5Z3  D R6Z4 D R6Z0 D R5Z0 ) SURF PLANE ;
*
 FLAN6 =( D ROZ3  R5Z3 D R6Z4  D ROZ4 D ROZ3 ) SURF PLANE
                      COUL VERT ;
 TUB6 = (D OZ3 ROZ3 D ROZ4 D OZ4 D OZ3) SURF PLANE COUL VERT ;
 AIREXT =( FLAN6 ET TUB6 ET FLAN5 ) coul  blan  ;
 AIRIN  =(FLAN1 ET FLAN2 ET FLAN3 ET BOBI ET FLAN4) coul vert ;
 TFLAN = AIRIN ET FER ET AIREXT ;ELIM .1 TFLAN ;
 PTAX = TFLAN POINT DROITE OO  ( 0 1. ) .01 ;
 TU1 = TFLAN ELEM APPU LARG PTAX ;
 TOUT = DIFF TFLAN TU1 ;
*
 C1 = (TOUT CONT ) COMPRIS ROZ4 ROZO ;
 C2 = C1 PROJETER (1 0 ) DROITE OO   OZ4  ;
 OZ4 = C2 POINT PROCHE OZ4 ;
 OO = C2 POINT PROCHE OO ;
 TU1 = DALLER C1 ( D 1 ROZO OO ) C2 (D OZ4 ROZ4 ) PLAN ;
 TUA = (COOR 2 TU1) POINT SUPER Z3 ;
 TUA = (TU1 ELEM APPU LARG TUA ) COUL BLAN ;
```

```
 TUB = (COOR 2 TU1) POINT INFER Z2 ;
 TUB = (TU1 ELEM APPU LARG TUB ) COUL  VERT  ;
 TUF = DIFF TU1 ( TUA ET TUB ) COUL ROUG ;
 AIRIN = (TOUT ELEM VERT ) ET TUB ;
 FER = (TOUT ELEM ROUG ) ET TUF ;
 AIREXT = ( TOUT ELEM BLAN) ET TUA ;
*  TRAC (AIRIN ET FER ET AIREXT )  FACE ;
*------------
*------------------ SHIFT TO  3D --------------------------------
*------------
 OPTION ELEM CUB8 ;
 SI ( EGA TYPEL 2 )
 OPTION DIME 3 ELEM CU20 ;
 FINSI ;
 ANG = 20. ;
*
*-----------  DEFINITION OF THE 2 DOMAINS -----------

*------  REDUCED POTENTIAL  VOLUME --------------------------
 DPHI = AIRIN VOLU 1 ROTA ANG (0 0 0 ) (0 1 0 ) ;

**------ TOTAL   POTENTIAL  VOLUME --------------------------
 DPSI = (AIREXT ET FER) VOLU 1 ROTA ANG (0 0 0 ) (0 1 0 ) ;
*
 ELIM .001  DPHI ; DPHI = REGE DPHI ;
  ELIM .001  DPSI ; DPSI = REGE DPSI ;
  ELIM .001  ( DPHI ET DPSI) ;

*---- ---------SEPARATION SURFACE  IDENTIFICATION -------------
* ----------  BELONGS TO REDUCED POTENTIAL  VOLUME ---------------
 SEP_PHI = INTE ( DPSI ENVEL ) ( DPHI ENVEL ) ;
*-------------  SPLITTING OF THE 2 DOMAINS ------------------
 DPSI = DPSI PLUS ( 0 0 0 ) ;
 SEP_PSI = SEP_PHI PLUS ( 0 0 0 ) ; ELIM .01 DPSI  SEP_PSI ;
*--------------------IRON-------------------------------------
 DFER = DPSI  ELEM ROUG ;
*
 VOLT = DPHI ET DPSI ;
 DAIR = DPSI ELEM BLAN  ;
 AA = DPHI POINT PLAN (0 0 0 ) ( 10 0 0 ) (0 0 10 ) .00001 ;
 AB =( DPHI ENVEL ) ELEM APPU STRIC AA ;
 AA = (AB ENVEL ) POINT DROITE  (0 0 0 ) ( 10 0 0 ) .00001 ;
 AA = ( (AB CONTOUR  ) ELEM APPU STRIC AA );
 deplacer vol1 tourner 90 ( 0 0 0 ) ( 1 0 0 ) ;
*
 OEXT = SEP_PHI POINT PROCHE (29. 0 0  ) ;
 BOBI = SEP_PHI POINT PLAN (0. 0. 0.) (10. 0. 0.) (0. 10. 0.)  .1 ;
 geomed = (dpsi et dphi ) point plan (0 0 0) ( 1 0 0) (0 1 0 ) .1 ;
 MU0 = 4. * PI * 1.E-4 ;
 RINT = 20. ;REXT =25. ; HAUT = 5. ;  J = 800. ;
 C1 = 0. 0. 0. ; ZP1 = 20. 0. 0. ;ZP2 = 0. 20. 0.  ;
 E  = Z1 * 4  ;
 UNIT = M  ;
 si ( ega UNIT  M) ;
 deplacer vol1  homo  .001  (0. 0. 0.) ;
 MU0 = 4. * PI  * 1.E-7 ;
 RINT = RINT/1000; REXT = REXT/1000 ; HAUT = HAUT/1000 ;
 ZP1 = ZP1/1000;ZP2 = ZP2 / 1000 ;J = J*1.E6 ;
 E = E /1000 ;
  finsi ;
 option echo 1 ;
*-------
*-- BIOT and SAVART COMPUTATION  of Hs  ON REDUCED POTENTIAL DOMAIN
*-------
 BS  = BIOT SEP_PHI  CERC C1 ZP1 ZP2  RINT REXT HAUT J  MU0 ;
 HS = BS / MU0 ;
```

```
  SI (NEG  GRAPH 'N')  ;
 OEIL = 1000 1000 500 ;
 TITRE  ' HS ON SEPARATION SURFACE ' ;
 VVVV = VECTEUR BS  1.e-3  'BX' 'BY' 'BZ'  ROUG ;
 TRAC OEIL  SEP_PHI VVVV ;
 option donn 5 ;
  FINSI ;
*---------------- FILLING OF A TABLE -----------------------
*-in this computation  there is only one isotropic material but it
* is shown how to coope with multi-material (even orthotropic)
*-------------------------------------------------------------
 TABB= TABLE ;
*
 FER1 = DFER  ELEM CU20      ;
 FER2 = DFER  ELEM PR15     ;
*
 KEVOL1 = H_B MU0         ;
 KEVOL2 = H_B MU0         ;
 OBFER1 = FER1  MODE THERMIQUE ISOTROPE ;
 OBFER2 = FER2  MODE THERMIQUE ORTHOTROPE  ;
 TABDEF  = TABLE ;
*----------------- isotropic material -----------------
 STN = TABLE ;
 STN.'EV1' = KEVOL1     ;
 TABDEF.OBFER1 = STN ;
*-------------   orthotropic material -------------------
 STN = TABLE ;
 DI1 = (0 1 0 ) ;DI2 = (1 0 1 ) ;
 STN.'EV1' = KEVOL1     ;STN.'DIR1' = DI1     ;
 STN.'EV2' = KEVOL2     ;STN.'DIR2' = DI2     ;
 TABDEF.OBFER2 = STN ;
 TABB.'TABNUSEC'= TABDEF;
*-------------------------------------------------------------
 TABB.'BIOT'=HS  ;
*-----------------   REDUCED POTENTIAL  AREA -------------
*---
 TABB.'DPHI' = DPHI ;
    imet = 0 ;
      si (ega imet 1 ) ;
 MOD1 = DPHI MODE THERMIQUE ISOTROPE ;
 MAT1 = MATE MOD1  'K' MU0 ;
 RIGC1 = CONDUC MOD1 MAT1 ;
 TABB.'RIGCON' = RIGC1 ;
      finsi ;
*
 TABB.'SEPPHI' = SEP_PHI ;
 TABB.'ORIG' = ORIG ;
 TABB.'MUAIR' = MU0;
*
*-------------------  TOTAL POTENTIAL AREA ---------------
*---------
 TABB.'DPSI' = DPSI ;
 TABB.'AIRPSI' = DPSI ELEM BLAN ;
     si ( ega imet 1 ) ;
 MOD2 = DAIR MODE THERMIQUE ISOTROPE ;
 MAT2 = MATE MOD2  'K' MU0 ;
 RIGC2 = CONDUC MOD2 MAT2 ;
 TABB.'RIGCPSI' = RIGC2 ;
      finsi ;

* TABB.'SEP_PSI' = SEP_PSI ;
*------------------ GENERAL BOUBDARY CONDITIONS ---------
 TABB.'BLOQUE' = BLOQUER 'T' GEOMED ;
*
*------ BOUNDARY CONDITION LINKED TO POTENTIAL JUMP  -----
*
 TABA = TABLE ;
 STN = TABLE ;TABA.1 = STN ;
```

```
 STN.'LGEO' = BANTI  ;STN.'MTYP' = 'TBLOQ' ;
 TABB.'POTSYM'= TABA  ;
 TABB.ISTEP = 1 ;
*-------------------- FIRST STEP  POTENTIALS COMPUTATION ----------
 POT_SCAL TABB     'SOLIN';
*-------------------------------------------------------------------
 SOL1= TABB.'POTENTIEL';
 PREF= 0. 0. 0. ;
*----- FIELD COMPUTATION         (B) -----------------------
 OBJM= MODL DPHI THERMIQUE ISOTROPE ;
 BA= CHAN CHPO (  GRAD OBJM  SOL1 ) OBJM   ;
 BA=  BA * -1. * MU0 ;
 BM= EXTR BA 'T,Z' OO ;
 option echo 0 ;
 SAUTER  5 LIGNES ;
 MESS '***********************************************************';
 MESS ' CIRCULAR COIL INTERNAL RADIUS'  RINT                 'mm  *';
 MESS '*      CROSS SECTION          ' ( REXT - RINT) '*' HAUT  'mm2';
 MESS '*      CURRENT DENSITY        '  J                   'A/mm2 *';
 MESS '***********************************************************';
 BS  = BIOT AA  CERC C1 ZP1 ZP2  RINT REXT HAUT J   MU0 ;
 MESS '*   INDUCTION  ON AXIS  WITHOUT IRON    (BIOT SAVART )';
 BS0= EXTR BS 'BZ' OO ;

 MESS '**************************************************************';
 MESS '*  AMPERE  mufer >> muo  ALL AMPERE*TURNS  IN THE GAP         ';
 MESS '*         GAP  = ' E  'mm                                     ';
 MESS '*       J * EP * HAUT = B/ MU0 * e                            ';
 MESS '**************************************************************';
 BTH = MU0 * J * (REXT - RINT ) * HAUT / E  ; BCAL = BM + BS0 ;
 MESS '**************************************************************';
 MESS ' BIOT  ET SAVART                          '  BS0       ;
 MESS ' TOTAL B   on AXIS ( z=0 )    EXPECTED      '  BTH       ;
 MESS '                              COMPUTED      '  BCAL      ;
*****************************************************************
   SI (NEG GRAPH 'N' ) ;
   TABLEG= TABLE ;TATT = TABLE ;
   TABLEG.1 = 'MARQ CROI' ;TATT.1=  'B_COIL' ;
   TABLEG.2 = 'MARQ PLUS' ;TATT.2=  'B_IRON' ;
   TABLEG.3 = 'MARQ CARR' ;TATT.3=  'B_TOTAL ' ;
   TABLEG.TITRE = TATT  ;
  BRBIO = ( EXCO BS 'BZ')  NOMC 'SCAL' ;
  BMRAY = EXCO (REDU BA AA) 'T,Z';
 EVBI = EVOL ROUG  CHPO BRBIO  'SCAL' AA ;
 EVBM = EVOL VERT  CHPO BMRAY  'SCAL' AA ;
 EVBT = EVOL TURQ  CHPO (BRBIO + BMRAY ) 'SCAL' AA ;
 TITRE ' BZ(R) for Z=0 ' ;
 DESS (EVBI ET EVBM ET EVBT ) LEGE TABLEG TITY TESLAS  ;
   FINSI ;
 RAP = ABS ((BCAL - BTH ) / BTH );
      SI ( RAP  > .01 ) ; ERREUR 5 ; FINSI ;
*----------------------------------------------------------------
*                   NON LINEAR  COMPUTATION
*----------------------------------------------------------------
 TABB.SOUSTYPE = THERMIQUE ;
 TABB.CRITERE =1.E-4 ;
 TABB.NITER   =1;
 TABB.'OME'= .99 ;
* ----------2 itérations only for the test ------------------------
 TABB.ITERMAX=2;
 TABB.NIVEAU =1;
*----------------------------------------------------------------
  MAG_NLIN  TABB  ;
*----------------------------------------------------------------
 SOL2 = TABB.'POTENTIEL';
 BA= CHAN CHPO (  GRAD OBJM  SOL2 ) OBJM   ;
 BA=  BA * -1. * MU0 ;
 BM= EXTR BA 'T,Z' OO ;
```

```
*---------------
    ITEST = 1 ;
            SI ( EGA ITEST 1 ) ;
 MESS '************************************************************';
 MESS '       VERIFICATION AU CENTRE                              ';
 MESS ' CONTRIBUTION FER  ATTENDUE  1.9496   CALCULEE ' BM        ;
*****************************************************************
 BAT = 1.9496 ;
*******************************************************************
 RAP = ABS ((BM - BAT ) / BAT );
      SI ( RAP  > .01 ) ; ERREUR 5 ; FINSI ;
*******************************************************************
   FIN ;
           FINSI ;
*------------
*------  CONTINUATION TILL CONVERGENCE OR UP TO 100 ITERATION ---
*----------
 TABB.ITERMAX=100 ;
*-------------------------------------------------------------------
 MAG_NLI3  TABB  ;
*-------------------------------------------------------------------
 SOL2 = TABB.'POTENTIEL';
 BA= CHAN CHPO (  GRAD OBJM  SOL2 ) OBJM   ;
 BA=  BA * -1. * MU0 ;
 BM= EXTR BA 'T,Z' OO ;
 BCAL = BM + BS0 ;
 MESS '************************************************************';
 MESS '      CHECK AT CENTER OF THE DEVICE                        ';
 MESS ' EXPECTED IRON CONTRIBUTION   1.9059  COMPUTED '  BM        ;
 MESS ' TOTAL  BZ  COMPUTED AT CENTER                '  BCAL      ;
 MESS '************************************************************';
 FIN;
```