
FORMA03 - Practical works of training “advanced Use”: Yield-point load of a perforated plate

Summarized:

This quasi-static 2D test in plane stresses makes it possible to illustrate on a simple case the relative questions with the elastoplastic modelization; it highlights the effects of structure, stress concentration, Yield-point load.

It is about a homogeneous rectangular plate, perforated in its center, consisted of an elastoplastic material with isotropic hardening, whose initial state is nonconstrained, which is subjected to a tension at its ends. One is interested in the elastoplastic solution in load.

The purpose of the test is to show the possibilities of modelization, the use of `STAT_NON_LINE` and the post-processing with `SALOME`.

The modelization A corresponds to computation with force imposed in elasticity. It illustrates the use of `STAT_NON_LINE` in a simplified configuration (linear purely elastic computation). It is also used as reference for the other modelizations.

The modelization B corresponds to computation with imposed force, of reference with behavior `VMIS_ISOT_TRAC`, and illustrates the use of the various parameters of `STAT_NON_LINE`, as well as the commands of examination.

The modelization C clarifies the procedure to carry out computation until the Yield-point load, by means of the control of the loading by a displacement.

The modelization D the identification of the parameters of a behavior (`VMIS_CIN1_CHAB`) illustrates, whose kinematic hardening is cancelled. The retiming of the parameters is done on the curve of

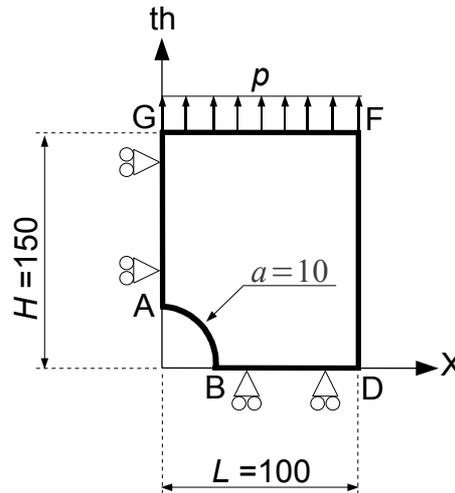
Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

tension used in the modelization B. One uses then these parameters B to carry out a computation similar to that of the modelization, with behavior VMIS_CIN1_CHAB.

1 Problem of reference

1.1 Geometry

It acts D `a rectangular plate, comprising a hole, modelled in 2D plane stresses. One models only one quarter of the plate thanks to symmetries. Dimensions are given in millimetres.



1.2 Boundary conditions and loadings

Conditions of symmetry

the plate is blocked according to Ox along the side AG and following Oy along the side BD .

Loading in imposed stress

It is subjected to a tension p according to Oy distributed on the side FG .

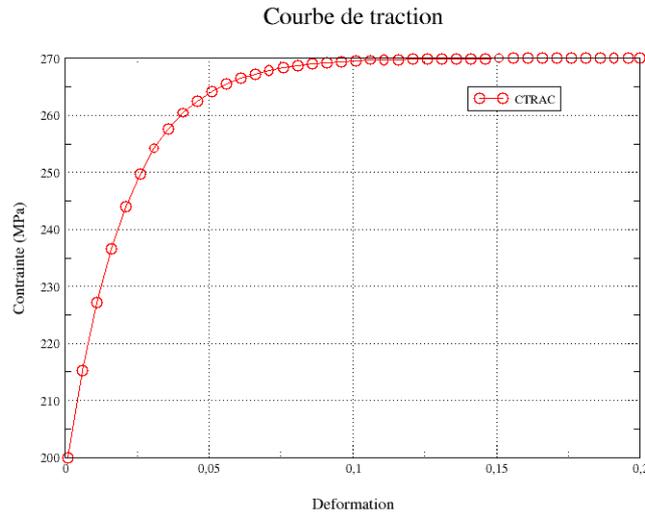
1.3 Properties of the materials

the behavior is elastoplastic of Von Mises, with isotropic hardening.

The elastic characteristics are:

- Young modulus $E = 200\,000\text{ MPa}$;
- Poisson's ratio $\nu = 0.3$;
- Elastic limit: 200 MPa ;

Hardening is deduced from the curve of tension defined by the following data (prolongation right constant `PROL_DROITE=' CONSTANT '`):



Epsilon	Sigma (Mpa)		Epsilon	Sigma (Mpa)
1.00000E-03	2.00000E+02		1.06000E-01	2.69626E+02
6.00000E-03	2.15275E+02		1.11000E-01	2.69709E+02
1.10000E-02	2.27253E+02		1.16000E-01	2.69773E+02
1.60000E-02	2.36630E+02		1.21000E-01	2.69823E+02
2.10000E-02	2.43964E+02		1.26000E-01	2.69862E+02
2.60000E-02	2.49694E+02		1.31000E-01	2.69893E+02
3.10000E-02	2.54168E+02		1.36000E-01	2.69917E+02
3.60000E-02	2.57659E+02		1.41000E-01	2.69935E+02
4.10000E-02	2.60382E+02		1.46000E-01	2.69949E+02
4.60000E-02	2.62506E+02		1.51000E-01	2.69961E+02
5.10000E-02	2.64161E+02		1.56000E-01	2.69969E+02
5.60000E-02	2.65451E+02		1.61000E-01	2.69976E+02
6.10000E-02	2.66457E+02		1.66000E-01	2.69981E+02
6.60000E-02	2.67240E+02		1.71000E-01	2.69986E+02
7.10000E-02	2.67850E+02		1.76000E-01	2.69989E+02
7.60000E-02	2.68325E+02		1.81000E-01	2.69991E+02
8.10000E-02	2.68696E+02		1.86000E-01	2.69993E+02
8.60000E-02	2.68984E+02		1.91000E-01	2.69994E+02
9.10000E-02	2.69209E+02		1.96000E-01	2.69996E+02
9.60000E-02	2.69384E+02		2.00000E-01	2.69996E+02
1.01000E-01	2.69520E+02			

2 elastic Reference solution

2.1 Solution

In elasticity, for an infinite **plate**, comprising a hole of diameter a , subjected to a loading P according to y ad infinitum, the analytical solution in plane stresses and polar coordinates (r, θ) is:

$$\sigma_{rr} = \frac{P}{2} \cdot \left[\left(1 - \left(\frac{a}{r} \right)^2 \right) - \left(1 - 4 \cdot \left(\frac{a}{r} \right)^2 + 3 \cdot \left(\frac{a}{r} \right)^4 \right) \cdot \cos(2\theta) \right] \quad (1)$$

$$\sigma_{\theta\theta} = \frac{P}{2} \cdot \left[\left(1 + \left(\frac{a}{r} \right)^2 \right) + \left(1 + 3 \cdot \left(\frac{a}{r} \right)^4 \right) \cdot \cos(2\theta) \right] \quad (2)$$

$$\sigma_{r\theta} = \frac{P}{2} \cdot \left[\left(1 + 2 \cdot \left(\frac{a}{r} \right)^2 - 3 \cdot \left(\frac{a}{r} \right)^4 \right) \cdot \sin(2\theta) \right] \quad (3)$$

In particular, at the edge of hole ($r=a$), one a:

$$\sigma_{\theta\theta} = p \cdot [1 + 2 \cdot \cos(2\theta)] \quad (4)$$

And along the axis x :

$$\sigma_{\theta\theta} = \sigma_{yy} = \frac{P}{2} \cdot \left[\left(1 + \left(\frac{a}{r} \right)^2 \right) + \left(1 + 3 \cdot \left(\frac{a}{r} \right)^4 \right) \right] \quad (5)$$

Numerically, for $P=10 \text{ MPa}$, and an infinite plate

Point	Component	MPa
A	SIGXX	-10
B	SIGYY	30

For a plate of finished **size**, the abacuses [bib1] make it possible to obtain the coefficient of stress concentration, and it is found that for a tension of 1 MPa , $SIGYY$ maximum is worth approximately 30.3 MPa at the point B .

2.2 Elastoplastic solution (Yield-point load)

In elastoplasticity, by a static approach in plane stresses, one can obtain a lower limit of the Yield-point load for a tape of width $2L$ finished and infinite length, comprising a hole of width $2a$ and subjected to an ad infinitum imposed stress p :

$$\bar{p}_{\text{lim}} = \frac{\sigma_y \cdot (L-a)}{L} \quad (6)$$

Here one obtains like limits lower Yield-point load: $\bar{p}_{\text{lim}} = 0.9 \times 270 = 243 \text{ MPa}$. (One takes here $\sigma_y = 270 \text{ MPa}$, because the Yield-point load is identical between a perfect elastoplastic material and a material whose curve of tension presents a horizontal asymptote to 270 MPa). In this test (in particular the modelization B), one would like to find, by an elastoplastic computation, an approximation of this Yield-point load, knowing that the analytical methods make it possible to know a lower limit of it. We will thus take the value \bar{p}_{lim} like reference.

2.3 Bibliographical references

- 1 Limit analysis of fissured structures and strength criteria. F. VOLDOIRE: Note EDF/DER/HI/74/95/26 1995
- 2 "Stress concentration factors", Peterson R.E., Wiley, 1974.

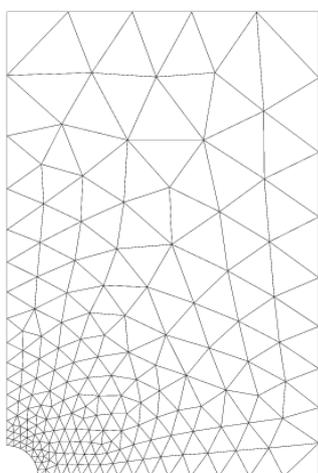
3 Modelization A

3.1 Characteristic of the modelization

Elastic design on a model in plane stresses (C_PLAN). The loading is defined in the § 3 . One charges until $p=10\text{ MPa}$.

3.2 Characteristics of the mesh

One uses a mesh which comprises 342 TRIA6 and 739 nœuds1Maillage¹.



MAILLAGE QUART DE PLAQUE TROUÉE

3.3 Quantities tested and results

One tests the value of the components stresses for the loading of 10MPa :

Standard	component of reference	Value	Tolerance
SIGM_NOEU – S_{IYY} in B	ANALYTIQUE	30 MPa	1.00%
SIGM_NOEU – S_{IXX} in A	ANALYTIQUE	– 10 MPa	2,00%
SIGM_NOEU – S_{IYY} in B	NON_REGRESSION	29.8 MPa	1e-6%
SIGM_NOEU – S_{IXX} in A	NON_REGRESSION	– 9,8 MPa	1e-6%

¹ obtained with version 6.3.0 of Salome, of light differences can and the appear according to the versions object computers

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

4 Modelization B

4.1 Characteristic of the modelization

One does three calculations on a model in plane stresses (C_PLAN):

- Elastic design: O N charges until $p=10\text{ MPa}$;
- Elastoplastic computation: O N charges until $p=230\text{ MPa}$;
- Elastoplastic computation then discharge: O N charges until $p=230\text{ MPa}$ then one discharges until $p=0$.

4.2 Characteristics of the mesh

One uses the same mesh as the modelization A which comprises 342 TRIA6 and 739 nodes.

4.3 Quantities tested and results

One tests the value of the components stresses for the elastic design (loading of 10MPa), one must find the same thing that in the modelization A, that is to say:

Standard	component of reference	Value	Tolerance
SIGM_NOEU – SIYY in B	Identical	AUTRE_ASTER A	
SIGM_NOEU – SIXX in A	Identical	AUTRE_ASTER A	

One tests the value of the components stresses and local variables for elastoplastic computation (to time corresponds to the loading of 230MPa):

Standard	component of reference	Value
SIGM_NOEU – SIYY in B	NON_REGRESSION	272.38393470555
SIGM_NOEU – SIXX in A	NON_REGRESSION	-213.68185288997
VARI_NOEU – V1 in B	NON_REGRESSION	0.35979129617121
VARI_NOEU – V2 in B	NON_REGRESSION	1.0
VARI_NOEU – V1 in A	NON_REGRESSION	3.5713883748082E-03
VARI_NOEU – V2 in A	NON_REGRESSION	1.0

One tests the value of the components stresses and local variables for elastoplastic computation with discharge (to time corresponds to the final unloading of 0):

Standard	component of reference	Value	Tolerance
SIGM_NOEU – SIYY in B	NON_REGRESSION	-298.10	1,00E-006%
SIGM_NOEU – SIXX in A	NON_REGRESSION	16.84	1,00E-006%
VARI_NOEU – V1 in B	NON_REGRESSION	0.364	1,00E-006%
VARI_NOEU – V2 in B	NON_REGRESSION	1.5	1,00E-006%
VARI_NOEU – V1 in A	NON_REGRESSION	3.55e-3	1,00E-006%
VARI_NOEU – V2 in A	NON_REGRESSION	0.0	1,00E-006%

5 Modelization C

5.1 Characteristic of the modelization

One does two calculations on a model in plane stresses (C_PLAN):

- Elastoplastic computation: one charges until $p=243 \text{ MPa}$;
- Elastoplastic computation: one does a calculation by control beyond the Yield-point load;

5.2 Characteristics of the mesh

One uses the same mesh as the modelization B which comprises 342 TRIA6 and 739 nodes.

5.3 Quantities tested and results

One tests the value of the Yield-point load for elastoplastic computation without control, compared to the analytical solution of the minimal limit and by NON-regression:

Standard	component of reference	Value	Tolerance
SIGM_NOEU – SIYY in G	ANALYTIQUE	243 MPa	1.00%
SIGM_NOEU – SIYY in G	NON_REGRESSION	243.04 MPa	1,00e-6%

One tests the value of the Yield-point load for elastoplastic computation with control (vertical displacement imposed of 6mm on the point G), compared to the analytical solution of the minimal limit, the solution without control and by NON-regression:

Standard	component of reference	Value	Tolerance
SIGM_NOEU – SIYY in G	ANALYTIQUE	243 MPa	1.00%
SIGM_NOEU – SIYY in G	NON_REGRESSION	243.55 MPa	1,00e-6%
SIGM_NOEU – SIYY in G	AUTRE_ASTER	243.04 MPa	0.30%
ETA_PILOTAGE in INST=6	ANALYTIQUE	243 MPa	1.00%
ETA_PILOTAGE in INST=6	NON_REGRESSION	243.51 MPa	1,00e-6%

6 Modelization D

6.1 Characteristic of the modelization

One do one on a model in plane stresses (C_PLAN) with identification of the parameters of model VISC_CIN1_CHAB by retiming.

6.2 Characteristics of the mesh

One uses the same mesh as the modelization A which comprises 342 TRIA6 and 739 nodes.

6.3 Quantities tested and results

One tests the value of the components stresses for a loading $p = 66.67 \text{ MPa}$:

Standard	component of reference	Value	Tolerance
SIGM_NOEU - SIYY in B	ANALYTIQUE	200 MPa	1,00%
SIGM_NOEU - SIXX in A	ANALYTIQUE	66.67 MPa	2,00%
SIGM_NOEU - SIYY in G	ANALYTIQUE	230 MPa	0,1%
SIGM_NOEU - SIYY in B	NON_REGRESSION	198 MPa	1.00e-6%
SIGM_NOEU - SIXX in A	NON_REGRESSION	65.47 MPa	1.00e-6%
SIGM_NOEU - SIYY in G	NON_REGRESSION	230 MPa	1.00e-6%

7 Implementation of the TP

7.1 Déroulement of the TP

It acts to conclude an elastoplastic computation by generating the geometry and the mesh using Salome, and the command file Code_Aster using Efficas.

This TP allows:

- To implement a standard nonlinear computation in Code_Aster: management of the loading, the materials, the behavior and the parameters of `STAT_NON_LINE` ;
- To understand and implement the notion of control;
- To implement a computation of retiming of the parameters of a nonlinear constitutive law;
- To make “advanced” postprocessings (to plot curves in particular);
- With regard to the test, script Python generating the geometry and the mesh in Salomé is in the file `datg` associated with the test.

7.2 Geometry

One will create a plane mesh of the quarter higher right it plates it, in quadratic elements, to have a sufficient accuracy. The mesh could be built with very simple Salomé of way. One can for example use `Sketcher` to define contours of the plate.

To launch modulus <code>GEOM</code> of Salomé;
New Entity/ 2D Sketch ;
It is simpler to start with the point B , of coordinates $(10,0)$. On the basis of B , for the arc of a circle, to use <code>Perpendicular</code> , and to define the angle and the radius. One obtains the point then A ;
To give the other points by their absolute coordinates;
To finish by <code>Closure Sketch</code> ;
One then obtains a closed contour (<code>Wire1</code>), on which one must build a face (<code>New Entity/Build/Face</code>).

The geometry of the plate is then complete. It is necessary to build the groups of the edges on which the boundary conditions (symmetries and loading) will be pressed:

To select <code>New Entity/Group/Create</code> ;
To select the type of entity geometrical (here line, <code>edge</code>) and to select edge directly in the chart window. Then it is necessary to click on <code>Add</code> . A number of object must then appear. One can change the name of the group before validating it by <code>Apply</code>
To build the following groups of edge thus: <code>left</code> for edge AG , <code>high</code> for edge GF , <code>right</code> for edge FD , <code>low</code> for edge DB and <code>hole</code> for the arc BA .

One can also create the groups leaning on the tops, they will be useful to create nodes groups (for postprocessing or control):

To select <code>New Entity/Group/Create</code> ;
To select the geometrical type of entity (here the top, <code>vertex</code>) and to select the top directly in the chart window. Then it is necessary to click on <code>Add</code> . A number of object must then appear. One can change the name of the group before validating it by <code>Apply</code> ;

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

To build the useful groups of tops thus: the five tops A , B , D , F and G .

7.3 Mesh

to carry out the mesh, one proposes to use Netgen:

To launch the modulus <code>MESH</code> of Salomé;
Mesh / Create Mesh ;
To select the Netgen algorithm then <code>1D-2D</code> to build the mesh (not to forget to select your geometry);
To add the assumption <code>Netgen Parameters 2D</code> ;
On this assumption, It is enough to select <code>Fineness/Very Fine</code> and to notch the box <code>Order Second</code> ;

One then obtains a mesh refined close to hole with large elements in the top of the plate. To refine this mesh, it is necessary to decrease `max Size` in the parameters of Netgen, while choosing for example `10` . If one wants to refine around hole, it is possible to use the cards of size in Netgen:

To publish the Netgen parameters;
To select the <code>Local mitre sizes</code> ;
To add the group hole with the button <code>One Edge</code> . It is then enough to modify the associated value. By decreasing it, the mesh will be refined on hole;
Not to forget to regenerate the mesh by <code>Mesh/Compute</code> ;

The groups are defined on the level of the geometry. One can import them in the mesh by `Mesh/Create groups from Geometry` or one will directly use the groups on the geometry in `Eficas`.

7.4 Postprocessings

For a nonlinear computation, `STAT_NON_LINE` leaves out of standard three fields (according to the options of keyword `ARCHIVAGE`) :

- The field of displacements to nodes `DEPL` ;
- Fields of the stresses with Gauss points `SO EF the _ELGA` ;
- The field of the local variables with Gauss points the `VARI_ELGA` ;

For postprocessing, one proposes besides calculating :

- The field of the stresses and the local variables to nodes (respectively `SIGM_NOEU` and `VARI_NOEU`). It is necessary to use for that `CALC_CHAMP` ;
- Equivalent stresses (Von Mises, Tresca, etc) with Gauss points with option `SIEQ_ELGA` of `CALC_CHAMP` ;

One proposes then several more advanced postprocessings:

Extraction of stress <code>SIYY</code> according to vertical displacement <code>DY</code> for the point G :
To extract the values from displacement at the point G as a result by means of command <code>RECU_FONCTION</code> on the field <code>DEPL</code> , component <code>DY</code> ;
To extract the values from the stress at the point G as a result by means of command

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

RECU_FONCTION on field SIGM_NOEU , component SIYY ;

To extract the values from the stress at the point G as a result by means of command RECU_FONCTION on field SIGM_NOEU , component SIYY ;

To print the function $SIYY=f(DY)$ (IMPR_FONCTION).

Extraction of stress SIYY on the lower edge BD :

Command MACR_LIGN_COUPE makes it possible to extract in an ARRAY the components from a field following a given path (keyword LIGN_COUPE). To apply the command to component SIYY of field SIGM_ELNO , to the path BD ;

To print the array resulting from MACR_LIGN_COUPE while filtering over last time and by displaying the columns of curvilinear abscisse ABSC_CURV and stress SIYY (IMPR_TABLE);

Extraction of stress SIYY on the lower edge BD :

Command MACR_LIGN_COUPE makes it possible to extract in an ARRAY the components from a field following a given path (keyword LIGN_COUPE). To apply the command to component SIYY of field SIGM_ELNO , to the path BD ;

To print the array resulting from MACR_LIGN_COUPE while filtering over last time and by displaying the columns of curvilinear abscisse ABSC_CURV and stress SIYY (IMPR_TABLE);

Extraction of the stresses on edge of hole:

Creation of a path with command INTE_MAIL_2D , one leans on edge of hole;

Command POST_RELEVE_T is used : one extracts (OPERATION=' EXTRACTION') the stresses $\sigma_{\theta\theta}$ along the path defined previously (keyword PATH) by projecting components SIXX , SIYY and SIXY of SIGM_ELNO in the polar coordinate system (keyword LOCATES); An array thus is produced;

To extract $\sigma_{\theta\theta}=f(s)$ with s curvilinear abscisse (ABSC_CURV) in the preceding array using RECU_FONCTION ;

In elasticity, one can compare this curve with the analytical reference (equation 4). One will create the analytical function:

Construction the formula (4) using a FORMULA which depends on the curvilinear coordinate s : SOLU=FORMULE (NOM_PARA=' , VALE=' $p * (1.+2.*\cos (2.*S/R))$) with $p=10\text{MPa}$ and $R=10\text{mm}$;

Interpolation of the formulation by the definition of a list of realities DEFI_LIST_REEL dividing the curvilinear coordinate of 0 with $s_{\max}=\frac{\pi}{2.R}$ then use of the command CALC_FONC_INTERP ;

Extraction of the resultant force at the top of the plate according to vertical displacement:

Computation of option FORC_NODA with CALC_CHAMP ;

To extract the values from displacement at the point G as a result by means of command RECU_FONCTION on the field DEPL , component DY ;

Creation of a path with command INTE_MAIL_2D , one leans on the top of the plate;

Command POST_RELEVE_T is used : one extracts (OPERATION=' EXTRACTION") L has resulting from displacements (keyword RESULTANTE) L E long of the path defined previously (keyword PATH) ; An array thus is produced;

To extract $DY=f(t)$ in the preceding array using RECU_FONCTION ;

Extraction of the parameter of control (modelization C):

To extract the values from the coefficient of control as a result by means of command RECU_FONCTION/ NOM_PARA_RESU on parameter ETA_PILOTAGE ;

To print the function $ETA_PILOTAGE=f(t)$ (IMPR_FONCTION).

8 Modelization A

8.1 Unfolding of the TP

It S" acts to conclude the elastic design by generating the command file Code_Aster using Efficas. The modelization is C_PLAN. A quarter of the plate is modelled. This modelization makes it possible to carry out a computation in elasticity for p energy until 10 MPa .

8.2 Elastic design

One will define in the command file, using Efficas, launched in Salomé:

To see the mesh with med format (LIRE_MAILLAGE);
To direct the mesh towards affected edge by the loading of Neumann (MODI_MAILLAGE/ORIE_PEAU_2D);
To affect the modelization formulates 2D plane stresses (AFFE_MODELE/C_PLAN);
To define material (DEFI_MATERIAU/ ELAS then AFFE_MATERIAU);
To affect the kinematical boundary conditions (conditions of symmetry on the quarter of plate by AFFE_CHAR_CINE for example): for that in EFICAS, one can select for each key word GROUP_MA a geometrical entity (group created from the geometry);
To affect the loading: distributed force (FORCE_CONTOUR) on the top of the plate. Simplest is to define a unit stress (FY=1.0), which one will then multiply by a function crawls;
To create a function crawls linear variable between 0. and 1000. to apply the mechanical loading (DEFI_FONCTION);
To create the temporal discretization using DEFI_LIST_REEL and of DEFI_LIST_INST ;
To use STAT_NON_LINE for the elastic design (COMP_INCR/RELATION=' ELAS ') with the list of time defined previously. If INST_FIN=10 is indicated. in keyword INCREMENT , one will have applied a force of formula 10MPa .

9 Modelization B

9.1 Unfolding of the TP

It acts to conclude elastoplastic computation with isotropic hardening given by a curve of tension such as the uniaxial stress tends towards a constant value (270 MPa).

There thus exists an Yield-point load for this structure whose lower limit is known: $p_{\text{lim}} > 243 \text{ MPa}$. In this modelization, one charges only until 230 MPa and one proceeds to an elastic return. The case of the Yield-point load will be treated in the elastoplastic modelization C

9.2 . Computation

9.2.1 Setting in initial data

One will define in the command file, using Efficas, launched in Salome. The command file is very similar to the modelization A, below, in fat, one indicates the differences :

To see the mesh with med format (<code>LIRE_MAILLAGE</code>);
To direct the mesh towards affected edge by the loading of Neumann (<code>MODI_MAILLAGE/ORIE_PEAU_2D</code>);
To affect the modelization formulates 2D (<code>AFFE_MODELE/ C_PLAN</code>) of them;
To see the curve of tension provided in the file forma03 B .21 (<code>LIRE_FONCTION</code>);
To define material (<code>DEFI_MATERIAU/ ELAS+ TENSION</code> then <code>AFFE_MATERIAU</code>);
To affect the kinematical boundary conditions (conditions of symmetry on the quarter of plate by <code>AFFE_CHAR_CINE</code> for example): for that in EFICAS, one can select for each key word <code>GROUP_MA</code> a geometrical entity (group created from the geometry);
To affect the loading: distributed force (<code>FORCE_CONTOUR</code>) on the top of the plate. Simplest is to define a unit stress (<code>FY=1.0</code>), which one will then multiply by a function crawls;
To create a first function crawls linear <code>SLOPE</code> varying between 0. and 1000. to apply the mechanical loading (<code>DEFI_FONCTION</code>);
To create the temporal discretization using <code>DEFI_LIST_REEL</code> (30 time step) and of <code>DEFI_LIST_INST</code> by activating the automatic cutting of time step: <code>ECHEC=_F (EVENEMENT=' ERREUR', ACTION=' DECOUPE',)</code> ;
To use <code>STAT_NON_LINE</code> for elastoplastic computation (<code>COMP_INCR/RELATION=' VMIS_ISOT_TRAC</code> ") with the list of time defined previously.

9.2.2 Computations to be realized

three different computations are proposed:

Elastic design:
If <code>INST_FIN=10</code> is indicated. in keyword <code>INCREMENT</code> of <code>STAT_NON_LINE</code> , by means of the <code>FONC_MULT</code> CRAWLS in <code>EXCIT</code> , one will have applied a force of formula 10 MPa ;
To check that one finds the same results as in the modelization A;

Elastoplastic computation in load:

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

If one indicates `INST_FIN= 23 0.` in keyword `INCREMENT` of `STAT_NON_LINE` , by means of the `FONC_MULT CRAWLS` in `EXCIT` , one will have applied a force of formula `230MPa` ;

One must have plasticization of structure because the resulting stress in part of structure is higher than the yield stress (which is worth `200MPa`);

Elastoplastic computation charges some and discharges:

One defines a new slope in the shape of hat:

1. $F=0.$ for $t=0$;
2. $F=230.$ for $t=230.$;
3. $F=0.$ for $t=300.$;

To define a news the list of times in ratio: `DEFI_LIST_REEL` (30 time step until $t=230.$, then 10 time step until $t=300.$) and of `DEFI_LIST_INST` by activating the automatic cutting of time step:

`ECHEC=_F (EVENEMENT=' ERREUR", ACTION=' DECOUPE',) ;`

Do a new calculation (new a `STAT_NON_LINE`) by taking `INST_FIN= 30 0.` in keyword `INCREMENT` of `STAT_NON_LINE` , and while not forgetting to use the new slope and the new list of time;

9.2.3 Results One

analyzes will be able to use modulus `VISU` of Salomé to visualize the fields of displacements, stresses and local variables . One will be able to then use the explorer of results (Modulus Stanley) in Salome, to visualize other fields (total deflections `EPSI_ELNO` for example), and to trace the evolution of `SIYY` along the segment `BD` .

Elastic case

some elements to be checked Here:

To check that one finds the same results as in modelization a: displacements and forced;

To check the indicator of plasticity (component `V1` of `VARI_ELGA`) . It must be null everywhere, idem for the cumulated plastic strain (component `V2`);

To observe the table of convergence: nombre of iterations of Newton;

Vary the temporal discretization (number of time step);

To compare the stress $\sigma_{\theta\theta}$ on edge of hole and to compare with the analytical solution (see § 11);

Plastic case charges some

With the discretization into 30 time step with the loading and the activation of the cutting of time step, it will not have there convergence. The algorithm of Newton fails. To try to make converge, you can exploit several parameters:

To increase the discretization of the loading (attention not to go too far, less than 100 steps) so that computation is not too long!);

To increase the maximum nombre of iterations of Newton which is worth 10 by defaults (`CONVERGENCE/ITER_GLOB_MAXI`);

To activate the linear search (RECH_LINEAIRE=_F ());
To increase the number of possible subdivisions of time step in the management of NON-convergence (DEFI_LIST_INST/ECHEC/SUBD_NIVEAU);
Combination of all the preceding techniques;

A combination which functions:

Discretization of the loading in 50 steps;
ITER_GLOB_MAXI = 20 ;
RECH_LINEAIRE activated;
Cut out up to five under-levels (DEFI_LIST_INST/ECHEC/SUBD_NIVEAU=5);

With this combination, one obtains the convergence in 358 steps (instead of the 30 initial ones, because of cuttings) and more than 4300 iterations.

One will be able post-to treat several ways according to the proposals of the §7.411 for example. Some results interesting to observe:

- With final moment of computation, for the maximum loading, one can notice on the isovaleurs of cumulated plastic strain, the localization of plastic strains (local variable v_1) in the vicinity of B . One will be able to use visualization with Gauss points to visualize the plastic strain equivalent cumulated to the places where it is calculated;
- For a loading lower than $66,7 MPa$, there is no plasticization;
- Until $230 MPa$, one is constantly in load.
- The maximum value of the criterion of Von Mises to Gauss points is always lower or equal to $270 MPa$, which shows that the solution checks the constitutive law well.

Let us observe the table of convergence to an unspecified step:

Time of computation: 2.297250000000e+02 - Level of cutting: 2

NEWTON		residual		residual	SEARCH.	LINE.	SEARCH.	LINE.	OPTION
ITERATION		RELATIF		ABSOLU	NB. ITER	COEFFICIENT	RHO	ASSEMBLY	
		RESI_GLOB_RELA		RESI_GLOB_MAXI					
0	X	1.81843E-04	X	4.64155E-01	0	- SANS OBJET	-	TANGENTE	
1	X	5.17708E-05	X	1.32145E-01	1	1.12982E+00			
2	X	2.67685E-05	X	6.83265E-02	1	1.46356E+00			
3	X	1.01270E-05	X	2.58491E-02	1	1.36817E+00			
4	X	4.14516E-06	X	1.05805E-02	1	1.58835E+00			
5	X	2.49245E-06	X	6.36199E-03	1	1.46980E+00			
6	X	1.35865E-06	X	3.46796E-03	1	1.68851E+00			
7		8.04731E-07		2.05407E-03	1	1.52519E+00			

We are at time 229.725 , one cut out twice the basic list and one converges in 8 iterations of Newton. One note:

- The linear search was not very expensive: an iteration only (by default $ITER_LINE_MAXI=3$). It is also seen that there is no linear search in prediction;
- Convergence is tested E on criterion $RESI_GLOB_RELA$. Without changing the default values of $STAT_NON_LINE$, one must thus have $RESI_GLOB_RELA$ lower than 1.0×10^{-6} to reach convergence ;
- The tangent matrix is calculated only in prediction, on several time step, one sees that it is always calculated with the first iteration. This corresponds well to the adjustment by default of $STAT_NON_LINE : REAC_INCR=1$ and $REAC_ITER = 0$

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

It is possible to require to STAT_NON_LINE to display more information: to follow the value of a degree of freedom (keyword SUIVI_DDL), or to ask to display the place where the convergence criterion is worst (keyword AFFICHAGE/INFO_RESIDU=' OUI'). This last adjustment allows, for example, to know which place of structure controls convergence. At the end of the transient of the statistical data are displayed:

```
Statistics on all the transient.
* Number of time step                : 358
* Nombre of iterations of Newton      : 4353
* Many factorizations of the matrix   : 358
* Many integrations of the behavior  : 8715
* Many resolutions K.U=F             : 4353

* Nombre of iterations of search linear : 4004

TEMPS CPU spent in the transient      : 2 m 29 S
of which time "wasted" in cuttings    : 55.640 S -> the list of time is effective to 62.8%
* Time assembly stamps                : 0.790 S
* Time construction second member     : 5.690 S
* total Time factorization stamps     : 1.390 S
* total Time integration behavior     : 2 m 4 S
* total Time resolution K.U=F        : 3.030 S
* different Time operations           : 14.120 S
```

We will detail all information but will not pass some note:

- For this one sees that the most consuming station is the integration of the constitutive law, well in front of the factorization and the resolution of the system. It is often the case in 2D, but it is especially related to the fact that one uses a noncomplete version of Newton. The matrix is factorized only once by time step (one also sees it on the number of factorizations: 358, like the number of time step);
- The initial list of time cut out in 5 0 time step was not most effective: the time wasted in computation because of failures of convergence and thus of the redécoupe of time step is of approximately a third of total time;

To improve convergence substantially, it is enough to activate complete Newton: NEWTON/REAC_ITER=1 . On 50 time step one gets the following results:

```
Statistics on all the transient.
* Number of time step                : 50
* Nombre of iterations of Newton      : 152
* Many factorizations of the matrix   : 152
* Many integrations of the behavior  : 202
* Many resolutions K.U=F             : 152

TEMPS CPU spent in the transient      : 5.330 S
* Time assembly stamps                : 0.220 S
* Time construction second member     : 0.520 S
* total Time factorization stamps     : 0.650 S
* total Time integration behavior     : 3.280 S
* total Time resolution K.U=F        : 0.110 S
* different Time operations           : 0.550 S
```

Besides the increase velocity of convergence (30 times faster), one observes any cutting of time step. One can even more coarsely cut out the list of times.

Plastic case charges then discharge with it

If one takes again the strategy allowing to carry out computation until the end in plasticity (i.e. until $p=230\text{ MPa}$), by minimizing temporal cutting (preceding exercise), one realizes that this strategy must be improved on the part discharges plastic because it does not converge with these adjustments.

It is pointed out that the discharge is done in an elastic way and creates an inelastic strain when the loading is null. So that computation converges, it is thus necessary to activate the elastic matrix in prediction (`NEWTON/PREDICTION=' ELASTIQUE'`).

10 Modelization C

10.1 Unfolding of the TP

It acts to conclude elastoplastic computation with isotropic hardening given by a curve of tension such as the uniaxial stress tends towards a constant value (270 MPa).

There thus exists an Yield-point load for this structure whose lower limit is known: $p_{lim} > 243 MPa$. In this modelization, one will show how to carry out computation beyond this Yield-point load thanks to control.

10.2 Detection of the Yield-point load

10.2.1 Put in data

One will define in the command file, using Efficas, launched in Salomé. The command file is almost identical to the modelization B. the only difference is that we propose here to activate the automatic management of the list of times, i.e. the temporal discretization is entirely managed by STAT_NON_LINE.

To see the mesh with med format (LIRE_MALLAGE);
To direct the mesh towards affected edge by the loading of Neumann (MODI_MALLAGE/ORIE_PEAU_2D);
To affect the modelization formulates 2D (AFFE_MODELE/ C_PLAN) of them;
To see the curve of tension provided in the file forma03c.21 (LIRE_FONCTION);
To define material (D EFI_MATERIAU/ ELAS+ TENSION then AFFE_MATERIAU);
To affect the kinematical boundary conditions (conditions of symmetry on the quarter of plate by AFFE_CHAR_CINE for example): for that in EFICAS, one can select for each key word GROUP_MA a geometrical entity (group created from the geometry);
To affect the loading: distributed force (FORCE_CONTOUR) on the top of the plate. Simplest is to define a unit stress (FY=1.0), which one will then multiply by a function crawls;
To create a function crawls linear SLOPE varying between 0. and 1000. to apply the mechanical loading (DEFI_FONCTION);
To activate the automatic management of time step with DEFI_LIST_INST : DEFAUTO=DEFI_LISTE_INST (DEFI_LIST=_F (METHODE=' AUTO', VALE= (0. , 50. , 243), PAS_MINI=1.e-6, PAS_MAXI=100.))
the list VALE gives three times of compulsory transition of the automatic list.
To use STAT_NON_LINE for elastoplastic computation (COMP_INCR/RELATION=' VMIS_ISOT_TRAC ") with the list of time defined previously.

10.2.2 Computations to be realized

One initially proposes to note “with the hand” the Yield-point load. The Yield-point load corresponds to the moment when a Gauss point reaches the equivalent value of Von Mises of approximately 270MPa . By analytical computation (see §2.25), one determined a lower

limit of the loading which causes a plasticization with this limiting value of 270MPa (around hole).

For that, to proceed as in the modelization B but made a loading beyond $p=230\text{MPa}$. Initially, a value of $p=245\text{MPa}$ is a good reference.

10.2.3 Results One

analyses will be able to use modulus VISU of Salomé to visualize the fields of displacements, stresses and local variables. One will be able to then use L" exploring of results (Modulus Stanley) in Salome, to visualize other fields (total deflections EPSI_ELNO for example), and to trace the evolution of SIYY along the segment BD .

It is seen that beyond a certain loading, the tangent matrix becomes singular: it is the sign which one reached the Yield-point load and thus a horizontal tangent on curve of tension. The code will try to cut out time step to go beyond this boundary point. By dichotomy (cutting of time step), it will approach the limiting value of loading. According to the meshes, it is found that $p_{lim} \approx 243\text{MPa}$.

The cause of difficult convergence is well the proximity of the Yield-point load. This is why it is necessary to subdivide time step. One can realize it by the value of the loading and curved force-displacement at the top of structure: one can note that for $p=240\text{MPa}$ the Yield-point load is not completely reached (not of horizontal asymptote) but that one approaches some. The isovaleurs of p approximately compared to the show a zone of concentration of plastic strain (comparable to line of sliding) 53° tilted of vertical, energy of the point B to flat rim. This corresponds rather well to the theory which says that the lines of sliding are tilted $54,44^\circ$ (see [bib2]). There is here of course an approximation of line of sliding which is in theory of thickness null. One can also note maximum vertical displacement according to Y point G . It is of approximately 5.7mm .

10.3 Computation of the Yield-point load by control

the best solution if one wishes to reach the limiting loading and to even go beyond (by the resolution of an incremental elastoplastic problem) is to use the control of the stress imposed by the displacement of a point. It is what one proposes here. One will be able to use for example displacement DY point G to control the stress σ_{yy} imposed on FG . One will increase it until 6mm for example (in preceding computation, one observed that the following maximum displacement Y of the point G was of approximately 5.7mm , 6mm is thus well beyond the Yield-point load). One will take a coefficient equal to 1. One will thus use a fictitious time t such as $\Delta t = \Delta U_Y(G) \times 1$. Thus time varies here enters 0 and 6s (to represent a displacement enters 0 and 6mm).

Note: in alternative to this kind of computation, a method of calculating of the Yield-point load is available in Code_Aster (in 3D , axisymmetric and plane strain): it uses a material of Norton-Hoff, quasi-incompressible elements and direct methods of limit analysis providing a framing of the Yield-point load (cf the documents [U2.05.04] and [R7.07.01]).

10.3.1 Put in data

One will define in the command file, using Efficas, launched in Salomé. L E beginning of the command file is identical to modelization b:

```
Reading of the mesh to med format ( LIRE_MAILLAGE );  
Direct the mesh towards affected edge by the loading of Neumann (
```

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

MODI_MALLAGE/ORIE_PEAU_2D);
Affect the modelization formulates 2D (AFFE_MODELE/ C_PLAN) of them;
Reading of the curve of tension provided in the file forma03c.21 (LIRE_FONCTION);
Define material (DEFI_MATERIAU/ ELAS+ TENSION then AFFE_MATERIAU);
Affect the kinematical boundary conditions (conditions of symmetry on the quarter of plate by AFFE_CHAR_CINE for example): for that in EFICAS, one can select for each key word GROUP_MA an E entity geometrical (group created from the geometry);
Create the temporal discretization using DEFI_LIST_REEL (10 time step until $t=4$) and of DEFI_LIST_INST by activating the automatic cutting of time step: ECHEC=_F (EVENEMENT=' ERREUR', ACTION=' DECOUPE',) ;
Modify the type of the loading (distributed force): in STAT_NON_LINE use EXCIT/TYPE_CHARGE=' FIXE_PILLO `` for the applied force;
Activation of control: keyword factor CONTROL : <ul style="list-style-type: none">•TYPE=' DDL_IMPO'' ;•COEF_MULT=1.0 ;•GROUP_NO=' G' ;•NOM_CMP=' DY' ;
To use STAT_NON_LINE for elastoplastic computation (COMP_INCR/RELATION=' VMIS_ISOT_TRAC ``).

10.3.2 Results Observing

in the file "message" the value of parameter ETA_PILOTAGE analyzes. One obtains in theory a good approximation of the Yield-point load (by higher value) than L" one will be able to compare with the analytical lower limit (243 MPa).

One will be able to plot, in poursuite, the curve forces resultant-displacement in G according to time. One should find on this curve the value of the Yield-point load given by "ETA_PILOTAGE".

One can visualize with Salomé, the deformed shape, the isovaleurs of stresses $SIYY$ and the cumulated equivalent plastic strain. At time 6 , one can notice on the isovaleurs of cumulated plastic strain, the localization of the strains in the vicinity of B .

11 Modelization D

This modelization makes it possible to illustrate the retiming of the coefficients material, and the use of these coefficients for the computation of the perforated plate. With this intention, one will use `astk` in external mode (out of Salomé) because it is necessary to define files other than the commands and the mesh. More precisely, it is a question of finding the parameters R_∞ and b of the behavior `VMIS_CIN1_CHAB` which are enough here to model only isotropic hardening. Here the principal equations of the model are summarized. The function threshold is worth:

$$F(\sigma, R, X) = (\tilde{\sigma} - X)_{eq} - R(p) \quad (7)$$

the isotropic function of hardening $R(p)$ is defined by:

$$R(p) = R_\infty + (R_0 - R_\infty)e^{-bp} \quad (8)$$

One defines also a model of normality (it is an associated model):

$$\dot{\varepsilon}^p = \dot{\lambda} \cdot \frac{\partial F}{\partial \sigma} = \frac{3}{2} \cdot \dot{\lambda} \cdot \frac{\tilde{\sigma} - X}{(\tilde{\sigma} - X)_{eq}} \quad (9)$$

$\tilde{\sigma}$ represents the deviator of the stresses and $()_{eq}$ the equivalent value within the meaning of von Mises. X is the isotropic tensor of hardening, it is written:

$$X = \frac{2}{3} \cdot C \cdot \alpha \quad (10)$$

With the law of evolution:

$$\dot{\alpha} = \dot{\varepsilon}^p - \gamma \cdot \alpha \cdot \dot{p} \quad (11)$$

the plastic multiplier:

$$\dot{p} = \dot{\lambda} = \sqrt{\frac{2}{3} \dot{\varepsilon}^p : \dot{\varepsilon}^p} \quad \text{and} \quad \begin{cases} \text{si } F < 0 \text{ ou } \dot{F} < 0 & \dot{\lambda} = 0 \\ \text{si } F = 0 \text{ et } \dot{F} = 0 & \dot{\lambda} \geq 0 \end{cases} \quad (12)$$

While choosing $C=0$, one does not reveal of kinematic hardening. One will be satisfied here to represent isotropic hardening, via the function $R(p)$, and to readjust the two parameters R_{inf} and b .

11.1 Unfolding of the TP

With regard to retiming, two command files are to be created: the file says "main" and the file says "slave".

The file says "main" (one will be able to take as example test ZZZZ159A) contains:

The definition of the parameters to be readjusted, their initial values and their limits, with following syntax:

```
parameters = [[ "R_INF", 150. , 200. , 500. ],
              [ "b", 11. , 10. , 100. ], ]
```

the experimental curve (one will take the curve of tension of the modelization A). Attention with the definition of this curve: one uses tables Numeric Python, in the form:

```
experiment = [Numeric.array ([[
```

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

```
1.00000E-03, 2.00000E+02], [  
6.00000E-03, 2.15275E+02], [  
1.10000E-02, 2.27253E+02], [  
1.60000E-02, 2.36630E+02], [  
2.10000E-02, 2.43964E+02], [  
2.60000E-02, 2.49694E+02], [  
3.10000E-02, 2.54168E+02], [  
3.60000E-02, 2.57659E+02], [  
4.10000E-02, 2.60382E+02], [  
4.60000E-02, 2.62506E+02], [  
5.10000E-02, 2.64161E+02], [  
5.60000E-02, 2.65451E+02], [  
6.10000E-02, 2.66457E+02], [  
6.60000E-02, 2.67240E+02], [  
7.10000E-02, 2.67850E+02], [  
7.60000E-02, 2.68325E+02], [  
8.10000E-02, 2.68696E+02], [  
8.60000E-02, 2.68984E+02], [  
9.10000E-02, 2.69209E+02], [  
9.60000E-02, 2.69384E+02], [  
1.01000E-01, 2.69520E+02], [  
1.06000E-01, 2.69626E+02], [  
1.11000E-01, 2.69709E+02], [  
1.16000E-01, 2.69773E+02], [  
1.21000E-01, 2.69823E+02], [  
1.26000E-01, 2.69862E+02], [  
1.31000E-01, 2.69893E+02], [  
1.36000E-01, 2.69917E+02], [  
1.41000E-01, 2.69935E+02], [  
1.46000E-01, 2.69949E+02], [  
1.51000E-01, 2.69961E+02], [  
1.56000E-01, 2.69969E+02], [  
1.61000E-01, 2.69976E+02], [  
1.66000E-01, 2.69981E+02], [  
1.71000E-01, 2.69986E+02], [  
1.76000E-01, 2.69989E+02], [  
1.81000E-01, 2.69991E+02], [  
1.86000E-01, 2.69993E+02], [  
1.91000E-01, 2.69994E+02], [  
1.96000E-01, 2.69996E+02], [  
2.00000E-01, 2.69996E+02], ]]
```

The names of the parameters of the array resulting from computation says "slave", which define the "numerical" curve calculated by simulation: computation

```
= [{"RESU1", "INST", "SIYY"},]
```

the call to MACR_RECAL (to specify UNITE_RESU for information on the parameters and their sensitivity, and possibly GRAPHIQUE , to visualize the variation enters the experimental curve and the various numerical curves evolving with the readjusted parameters: RESU

```
=MACR_RECAL (UNITE_ESCL  
=3          , UNITE_RESU  
=91          , RESU_EXP  
=EXPERIENCE          , LIST_PARA  
=PARAMETRES          , RESU_CALC  
=CALCUL          , GRAPHIQUE  
= _F (UNITE=90),);
```

The file says "slave" making it possible to simulate, for a set of parameters given, the simple traction test with Code_Aster . With this intention, one will be able to use SIMU_POINT_MAT which makes it possible to simulate a test with strain (or stress) imposed on a material point. It is enough to provide the parameters of the material (here, CIN1_CHAB), the function defining the evolution of the strain imposed according to times, and the urgent of computation.

The parameters to be readjusted must be suffixed by two "_": #
parameters to readjust (the provided values do not have importance for retiming) R_

```
INF = 200; b_  
_ = 20.0; MY  
  
=DEFI_MATERIAU (ELAS=_F (E=200000.0, NU =0.3, ), CIN  
1_CHAB=_F (R_0=200., R_  
I=R_INF, B=  
b, C_  
I=0., G_ 0=1., );
```

The function defining the evolution of the strain imposed according to time can be written here simply: EPSIMP

```
=DEFI_FONCTION (NOM_PARA=' INST', VALE= (0.0, 0.0, 1. , 1.))
```

List of computation moments between 0. and 0.2: LI

```
1=DEFI_LISTE_REEL (DEBUT=0, INTERVALLE=_F (JUSQU_A=0.2, ..., )) It
```

remains to call on SIMU_POINT_MAT . Result is an array containing, for each calculated time, the 6 components of the strains, the stresses, and the local variables. This is why in the "main" file, the name of array RESULTAT is RESU 1, and the parameters are EPYY , SIYY , which makes it possible to build a curve of tension (forced according to the total deflection): RESU

```
1=SIMU_POINT_MAT ( COMP_INCR=_F (RELATION=' VMIS_CIN1_CHAB', ITER  
INTE_MAXI=100, ), MATER  
=MATER, INCREMENT  
=_F (LIST_INST=LI1, ), NEWTON  
=_F (REAC_ITER=1, ), EPSI  
_IMPOSE=_F (EPYY=EPSIMP, ), ); Before
```

launching optimization, it is useful to check that this computation only functions. To launch optimization, while associating with this slave file unit 3. (computation can take with 100s). 200s To observe convergence and the values of the parameters in file 91. One will carry out then the computation of the perforated plate, as for the modelization A, with behavior VMIS_CIN1_CHAB , and the p readjusted arameters. One will be able to check that one gets results similar to those of modelization A. Synthèse

12 of the results This

test allows to show how to carry out the computation of an elastoplastic structure and his examination, and in particular to highlight the benefit to use control for a problem of Yield-point load. One can retain of this test some ideas: Even

- apart from a perfect elastoplastic behavior, it can exist an Yield-point load. It is the case with all real curves of tension. It is then necessary to adapt the method of resolution to the mechanical solution and for example to use control;
- Cutting in small increments of load is often necessary to integrate the behavior model correctly. That can also help with convergence, it is thus advised to use the automatic recutting of time step;
- The linear search can be used to help with convergence, as well as the automatic subdivision of time step. In the event of discharge, the elastic prediction is an effective solution.