FORMA03 - Practical works of the formation “advanced Use”: load limits of a perforated plate

Summary:

This test $2D$ in plane constraints quasi-static allows to illustrate on a simple case the relative questions with elastoplastic modeling; it highlights the effects of structure, of limiting load, stress concentration.
It is about a homogeneous rectangular plate, perforated in its center, consisted of an elastoplastic material with isotropic work hardening, whose initial state is nonconstrained, which is subjected to a traction at its ends. One is interested in the elastoplastic solution in load.
The objective of the test is to show the possibilities of modeling, the use of the order `STAT_NON_LINE` and postprocessing with the platform `Salomé-Meca`.

Modeling A corresponds to calculation with force imposed in elasticity. It illustrates the use of the order `STAT_NON_LINE` in a simplified configuration (linear purely elastic calculation). It is also used as reference for other modelings.

Modeling B corresponds to calculation with imposed force, of reference with the behavior `VMIS_ISOT_TRAC`, and the use illustrates of the various parameters of the order `STAT_NON_LINE`, as well as the orders of examination.

Modeling C clarifies the procedure to carry out calculation until the limiting load, by using the piloting of the loading by a displacement.

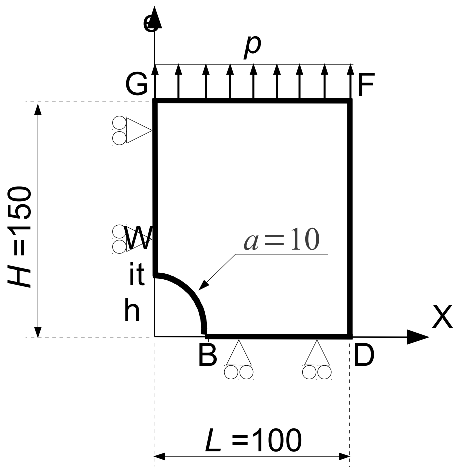
Modeling D is identical to modeling C except that it uses a loading follower.

Contents

1 Problem of reference

1.1 Geometry

It is about a rectangular plate, comprising a hole, modelled in $2D$ plane constraints. 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 .

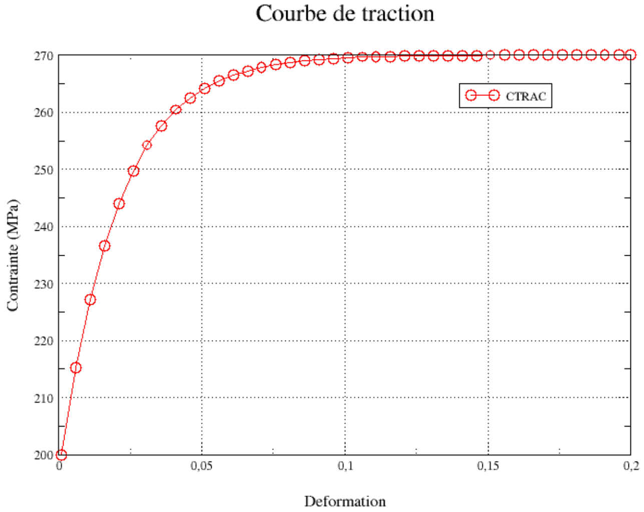
Loading in imposed constraint
It is subjected to a traction p according to Oy distributed on the side FG .

1.3 Properties of materials

The behavior is elastoplastic of Von Mises, with isotropic work hardening.

Elastic characteristics are:
- Young modulus $E=200\,000\,MPa$;
- Poisson's ratio $\nu=0.3$;
- Elastic limit: $200\,MPa$.

Work hardening is deduced from the traction diagram defined by the following data (prolongation constant right `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 Reference solution

2.1 Elastic solution

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

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

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

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

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

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

And along the axis x :

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

Numerically, for $P=1\,MPa$, and for an infinite plate, one has

Not	Component	Calculation	MPa
A	$SIXX$	$\sigma_{\theta\theta}(r=a,\theta=\pi/2)$	– 1
B	$SIYY$	$\sigma_{\theta\theta}(r=a,\theta=0)$	3

For a plate of dimension **finished**, the abacuses [bib1] make it possible to obtain the coefficient of stress concentration, and one finds that for a traction of $1\,MPa$, $SIGYY$ maximum is worth approximately $3.03\,MPa$ at the point B .

2.2 Elastoplastic solution (load limits)

In elastoplasticity, by a static approach in plane constraints, one can obtain a terminal supérieure of the load limits for a band of width $2L$ finished and infinite length, comprising a hole of width $2a$ and subjected to an ad infinitum imposed constraint p :

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

Here one obtains as limits supérieure of the limiting load: $\bar{p}_{\lim}=0.9\times270=243\,MPa$. (One takes here $\sigma_y=270\,MPa$, because the limiting load is identical between an elastoplastic material perfect and a material whose traction diagram presents a horizontal asymptote to $270\,MPa$). In this test (in particular modeling B), one would like to find, by an elastoplastic calculation, an approximation of this limiting load, knowing that the analytical methods make it possible to know a terminal of it supérieure. We will thus take the value $\bar{p}_{\lim}$ like reference.

2.3 Bibliographical references

[1] Analysis limits fissured structures and criteria of resistance. F. VOLDOIRE: Note EDF/DER/HI/74/95/26 1995

[2] "Stress concentration factors", Peterson R.E., Wiley, 1974.

3 Implementation of the TP

3.1 Unfolding of the TP

It is a question of concluding the elastic design by generating the geometry, the grid and the command file AsterStudy using the platform SalomEMeca.
This TP allows:

- To implement a standard non-linear calculation in the module AsterStudy : management of the loading, materials, the behavior and the parameters of `STAT_NON_LINE` ;
- To understand and implement the concept of piloting;
- To make "advanced" postprocessings (to plot curves in particular).

3.2 Geometry

One will create the plane face of the quarter higher right DE the plate.

To launch the module `Geometry`.

Principal stages to build this geometry are the following ones :

- To define contours of the plate, one can, for example, to use the tool " `Sketcher` " (Finely `New Entity → Lowic → 2D Sketch`) . It is simpler of commencer by not B of coordinates $(10,0)$. On the basis of B , for the arc of a circle, to use `Standard element (Withrc)` and `Destination (Direction/Perpendicular)` , and to define the ray `10` and L' angle and the ray `90°`. The point is obtained A . Then to use `Standard element (Line)` and Donner other points (G , F , D) by their absolute coordinates. To finish by `Closure sketch`.
- A closed contour is then obtained `(Sketch_ 1 )` on which one must build a face (Menu `New Entity → Build → Face`) . The geometry of the plate is then complete.
- To build useful groups for calculation. Here one builds 5 groups D be edges on which the boundary conditions (symmetries and loading) will be pressed: `left` for the edge AG , `high` for the edge GF and `low` for the edge BD , `right-hand side` for the edge FD and `hole` for the arc AB . Menu `New Entity → Group → Create Group` : To select the geometrical type of entity (here the line, `edge`) and to select the edge directly in the chart window, E nsuite to click on `Add` , U N number of object must then appear. One can change the name of the group before L E to validate by `Apply` .
- One can also create the groups nodes, which will be useful for postprocessing or piloting (Menu `New Entity → Group → Create Group`): five groups of top A , B , D , F and G .

3.3 Grid

One will create a grid plan of the quarter higher right it plates it, in quadratic elements, to have a sufficient precision.

To launch the module `Mesh`.

Principal stages for to generate the grid are the following ones :

- To build grid (Menu `Mesh → Create Mesh`) . To select geometry to be netted `Face_1` , then to choose `With lgorithm → NETGEN 1D-2D` while adding `H ypothe located → NETGEN 2D Parameters` . On this assumption, to select `Fineness → Fine` and coachman the box `Second Order` before `Apply` .
- Calculer it grid (Finely `Mesh → Compute`) . A window of information of grid must appear, and ON obtains then a grid refined close to the hole with large elements in the top of the plate.
- To refine this grid, to click on the right on the grid and to choose `Mesh edict`, then to publish the parameters `NETGEN 2D Parameters` :

  - One can to decrease `Max Size` while choosing for example 10 .
  - If one wants to refine around the hole, one can in the mitre `Room sizes` hasjouter the group `hole` AB with the button `One Edge` , and then IL is enough to modify the associated value. By decreasing it (for example 2), the grid will be refined around hole.
- Calculer it grid (Finely `Mesh → Compute`) .
- To make pass the grid of linear to quadratic: " `Modification - > Convert to/from quadratic` ".
- To create Lbe groups DE e-mailbe geometrical correspondents with the group (Menu `Mesh → Create Groups FRomanian Geometry`) . Selectionner all geometrical groups.
- Exporter it grid with format MED.

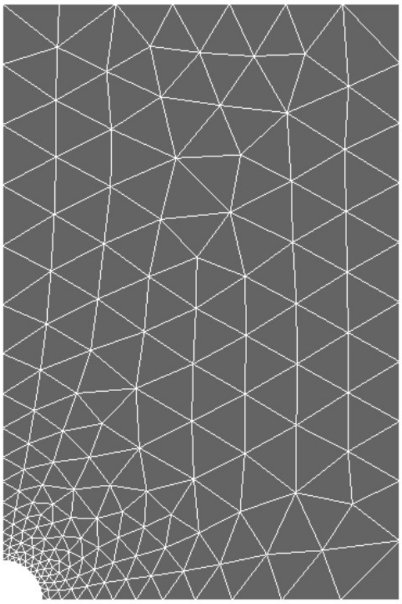
4 Modeling A

4.1 Characteristics of modeling

Elastic design of one quarter of the plate on a model in plane constraints (C_PLAN). The loading is defined in the § 1.2 . One charges until $p=10\,MPa$.

4.2 Characteristics of the grid

A grid is used quadratic.



4.3 Elastic design with STAT_NON_LINE

It is a question of concluding the elastic design by generating the command file AsterStudy using the platform Salomé-Meca. Modeling is C_PLAN. One must find the same results as by using the order MECA_STATIQUE.

4.3.1 Realisation of calculation

To launch the module AsterStudy.
Then in left column, to click on the mitre View box .
One defines the command file of the calculation case (to click on the right with CurrentCase and to choose Add Internship).
Foot-note: to add orders by the menu Commands → All show .

Principal stages for the creation and the launching of the calculation case are the following ones:

- To see the grid with format MED: Order LIRE_MAILLAGE .
- To direct Lhas normal edge on which the loading of traction will be applied : Category Mesh / Order MODI_MAILLAGE/ORIE_PEAU_2D in affecting the group hauT in GROUP_MA. One keeps the same name of the grid while using reuse.
- To define the finite elements used: Order AFFE_MODELE for to affect the phenomenon MECHANICS and modeling in plane constraints 2D (C_PLAN) with all the elements.
- To define material: Order DEFI_MATERIAU. To choose ELAS and to seize the values of the Young modulus and the Poisson's ratio.

- To affect material with all the elements : Order AFFE_MATERIAU.
- To affect conditions with limiting kinematics : Order AFFE_CHAR_MOVIES / MECA_IMPO for Symetry on the quarter of plate (groups left and low).
- To affect the loading : Order AFFE_CHAR_ MECA/ FORCE_CONTOUR for the force distributed on the top of the plate. Simplest is to define a unit stress (FY=1.0) , that one will multiply then by a function crawls with time.
- To create a function crawls linear $f=t$ for to multiply the mechanical loading unit : Order DEFI_FONCTION. For example, it variE enter $(0.,0.)$ and $(1000.,1000.)$
- To create the temporal discretization usingC MommandeS DEFI_LIST_REEL and DEFI_LIST_INST. For example, one can determine the end of moment with 10s to correspond the loading to $10\,MPa$.
- To calculate the elastic evolution: Order STAT_NON_LINE. One puts BEHAVIOR/ RELATION= ' ELAS ', the list of moment defined previously in INCREMENT, materials in CHAM_MATER, MODEL and also them boundary conditions and the loading (LOAD + FONC_MULT) in EXCIT.

For launchR the calculation case, in left column, to click on the mitre History View .

4.3.2 PostprocessingS results hasvec Aster_Study

For a non-linear calculation, the order STAT_NON_LINE fate out of standard three fields (according to the options of the keyword FILING :

- The field of displacements with the nodes DEPL ;
- The field of the constraints at the points Gauss SIEF _ELGA ;
- The field of the internal variables at the points of Gauss VARI_ELGA.

For postprocessing, one proposes besides calculating using CALC_CHAMP with reuse :

- The field of the constraints with the nodes (SIGM_NOEU) by option CONSTRAINT.
- Equivalent constraints (Von Mises, Tresca, etc) at the points of Gauss, SIEQ_ELGA by option CRITERIONS.

One proposes to print the results with the format MED with IMPR_RESU in order to visualize them in Results however Paravis.

One proposes then several postprocessings more evolved (facutatif) :

Extraction of the constraint SIYY according to vertical displacement DY for the point G :

- To extract vertical displacement DY at the point G : order RECU_FONCTION by using it result, and choosing the field DEPL and component DY .
- To extract L forced SIYY at the point G : order RECU_FONCTION by using it result, and choosing the field SIGM_NOEU E T component SIYY .
- To print the function $SIYY=f(DY)$: Order IMPR_FONCTION with the format XMGRACE (keyword CURVE → FONC_X and FONC_Y) .

Extraction of the constraint SIYY on the lower edge BD :

- C alcule R L E field of the constraints by elements to the nodes (SIGM_EL NO): order FORCED CALC_CHAMP/.
- To extract a table from constraint SIYY at the certain points on the edge BD : order MACR_LIGN_COUPE allows to extract in one table components of a field following a way given (keyword LIGN_COUPE). To apply the order to the component SIYY field SIGM_ELNO , with 10 points on the way BD (by giving the coordinates of the points B and D) .
- To print a curve starting from a table: order IMPR_TABLE for I mprimer with the format XMGRACE the table the preceding one while filtering over the last moment (for example, FILTER/ NOM_PARA = 'INST', VALE= 10). The parameters with axes X and Y of the curve are defined by NOM_PARA : curvilinear X-coordinate (ABSC_CURV) and constraint (SIYY) .

Extraction of the constraints on Bord of the hole AB :

- Ewill xtraiRe Lhas constraint $\sigma_{\theta\theta}$ along the edge of the hole: order `MACR_LIGN_COUPE`. In the keyword `LIGN_COUPE`, one specifies `TYPE=ARC` and the reference mark `POLAR`. ON defines 10 points on the edge of the hole by giving the coordinates DU not of departure B and center O . One filter at the last moment `INST = 10` and product thus a table .
  ***Notice** : in the reference mark `POLAR` in 2D, The significance of the components is: `DX` ray r , `DY` angle θ . Thus for $\sigma_{\theta\theta}$ one has `NOM_CMP = IFYY` .*
- To extract the function $\sigma_{\theta\theta}=f(s)$ with the curvilinear X-coordinate $s=\theta R$ ($s\in[0, R\pi/2]$) : order `RECU_FONCTION`. One defines `PARA_X` with the curvilinear X-coordinate (`ABSC_CURV`) and `PARA_Y` with $\sigma_{\theta\theta}$ (`IFYY`).

- In elasticity, one can compare it digital result with the analytical reference (equation 4).
  - One can to create formula analytical: order `FORMULA`. It depends on the curvilinear coordinate S (`NOM_PARA='/VALE=' p* (1.+2.*cos (2. *S/R))'` with $p=10\text{MPa}$ and $R=10\text{mm}$).
  - To create a list of S values reality S of S from 0 with $s_{\max}=\pi R/2$: Order `DEFI_LIST_REEL`.
  - Interpolation of formula starting from the list of S : order `CALC_FONC_INTERP` .
- C ommande `IMPR_FONCTION` to print with the format `XMGRACE` L be curves of two function S: C it resulting from the result digital $\sigma_{\theta\theta}=f(s)$ and that formula analytical .

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

- Calculation of the option `FORC_NODA` with the order `CALC_CHAMP/ FORCE` .
- To extract displacement vertical at the point G : order `RECU_FONCTION` by using it result, and choosing the field `DEPL` , component `D Y` .
- E xtraire of the vertical resulting force on the edge `high` plate: order `MACR_LIGN_COUPE`. One chooses `RESULT` and `NOM_CHAM=FORC_NODA` . In the keyword `LIGN_COUPE` , O N defines points on which one wishes to calculate the resultant: to give the coordinates of G and F , the number of the points, and `RESULTANTE=DY` . One produces then a table.
- To extract the function of the resultant vertical according to time $Resultante_{DY}=f(t)$: order `RECU_FONCTION` .
- To print function $Resultante_{DY}=f(DY)$: C ommande `IMPR_FONCTION` with the format `XMGRACE` .

4.4 Sizes tested and results

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

Component	Type of reference	Value	Tolerance
`SIGM_NOEU` – $SIYY$ in B	`ANALYTICAL`	$30\,MPa$	1,00%
`SIGM_NOEU` – $SIXX$ in A	`ANALYTICAL`	$-10\,MPa$	2.00%

5 Modeling B

5.1 Characteristics of modeling

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

- Elastic design: one charges until $p=10\,MPa$;
- Elastoplastic calculation: one charges until $p=230\,MPa$;
- Elastoplastic calculation then discharge: one charges until $p=230\,MPa$ then one discharges until $p=0$.

5.2 Characteristics of the grid

One uses the same grid as modeling A (who comprises 315 TRIA6 and 686 nodes).

5.3 Elastoplastic calculation with STAT_NON_LINE

It is a question of concluding elastoplastic calculation with isotropic work hardening given by a traction diagram such as the uniaxial constraint tends towards a constant value ($270\,MPa$).
There thus exists a limiting load for this structure of which a terminal supérieure is known $p_{\lim}<243\,MPa$. In this modeling, one charges only until $230\mathrm{MPa}$ and one proceeds to an elastic return. The loading case limits will be treated in LE following paragraph.

5.3.1 Preparation of the command file

To launch the module AsterStudy .
Then in left column, to click on the mitre View box .
One defines the command file of the calculation case (to click on the right with CurrentCase and to choose Add Internship).
Foot-note: to add orders by Menu Commands → All show .

One defines the command file of the calculation case. The command file is very similar to modeling the preceding one , below, in fat, the differences are indicated:

- To see the grid with format MED: Order LIRE_MAILLAGE .
- To direct Lhas normal edge on which the loading of traction will be applied : Category Mesh / Order MODI_MAILLAGE/ORIE_PEAU_2D in affecting the group hauT in GROUP_MA. One keeps the same name of the grid while using reuse.
- To define the finite elements used: Order AFFE_MODELE for to affect the phenomenon MECHANICS and modeling in plane constraints 2D (C_PLAN) with all the elements.
- **To see the traction diagram provided in the file forma03b.21** : Order LIRE_FONCTION / NOM_PARA = 'EPSI'.
- D**to éfinir material** : Order DEFI_MATERIAU/ ELAS and TRACTION (to affect traction diagram).
- To affect material with all the elements : Order AFFE_MATERIAU .
- To affect conditions with limiting kinematics and the loading : Order AFFE_CHAR_ MOVIES / MECA_IMPO for Symetry on the quarter of plate (groups left and low).
- To affect the loading : Order AFFE_CHAR_ MECA/ FORCE_CONTOUR for the force distributed on the top of the plate. Simplest is to define a unit stress (FY=1.0) , that one will multiply then by a function crawls with time.
- To create a function crawls linear $f=t$ for to multiply the mechanical loading unit : Order DEFI_FONCTION. For example, it variE enter $(0.,0.)$ and $(1000.,1000.)$
- **To create the temporal discretization** usingC MommandeS DEFI_LIST_REEL and DEFI_LIST_INST. For example, one can determine 30 pas de time until 300s for to

correspond the loading maximum with $300\,MPa$. In DEFI_LIST_INST, activer the automatic cutting of the step of time: FAILURE/EVENEMENT= ' ERROR ' and WithCTION= ' CUTTING '.

- **To calculate the evolution elastoplastic** : Order STAT_NON_LINE. One puts BEHAVIOR/ RELATION= ' VMIS_ISOT_TRAC ', the list of moment defined previously in INCREMENT, materials in CHAM_MATER, MODEL and also them boundary conditions and the loading (LOAD + FONC_MULT) in EXCIT.

5.3.2 Calculation rubber band

If one indicates INST_FIN = 10S daNS the keyword INCREMENT order STAT_NIN_LINE, one will have applied a force well of $10\,MPa$, what is equivalent strictly to the elastic case.
Here some elements to be checked:

- To check that one finds the same results as in modeling the preceding one : displacements and constraints.
- To check the indicator of plasticity (VARI_ELGA) . It must be null everywhere, idem for the cumulated plastic deformation (EPSP_ELGA).
- To observe the table of convergence: iteration count of Newton.
- Vary the temporal discretization (many steps of time).
- To compare the constraint $\sigma_{\theta\theta}$ on the edge of the hole and to compare with the analytical solution.

5.3.3 Calculation elastoplastic in load

If one indicates INST_FIN = 230S Dyears the keyword INCREMENT order STAT_NIN_LINE, one will have applied a force well of $230\,MPa$.
One must have plasticization of the structure because the resulting constraint in part of the structure is higher than the yield stress (which is worth $200\mathrm{MPa}$).

With the discretization in 30 pas de time of the loading and the activation of the cutting of the step of time, there will be no 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 100 pas) so that calculation is not too long!).
- To increase the maximum iteration count of Newton which is worth 10 by defaults (STAT_NON_LINE/ CONVERGENCE/ITER_GLOB_MAXI).
- To activate linear research (STAT_NON_LINE/ RECH_LINEAIRE).
- To increase the number of possible subdivisions of the step of time in the management of not-convergence (DEFI_LIST_INST/ECHEC/SUBD_NIVEAU).
- Combination of all the preceding techniques.

Here Ucombination which does not function in our case :

- Discretization of the loading in 50 pas;
- STAT_NON_LINE/ CONVERGENCE/ITER_GLOB_MAXI = 20 ;
- Activated linear research;
- Cutting up to five under-levels (DEFI_LIST_INST/ECHEC/SUBD_ NIVEAU=5).

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

Some results interesting to observe:

- At the final moment of calculation, for the maximum loading, one can notice on the isovaleurs of cumulated plastic deformation, the localization of the plastic deformations (variable internal V1) in the vicinity of B . One will be able to use visualization at the points of Gauss to visualize the plastic deformation 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 at the points of Gauss is always lower or equal to $270\,Mpa$, which shows that the solution checks the law of behavior well.

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

```
 Moment of calculation: 2.297250000000e+02 - Level of cutting: 2

-------------------------------------------------------------------------------------------------------------
|     NEWTON      |      RESIDUE      |      RESIDUE      |  RESEARCH.  LINE.  |  RESEARCH.  LINE.  |     OPTION      |
|    ITERATION    |      RELATIVE     |      ABSOLUTE      |     NB. ITER      |  COEFFICIENT    |    ASSEMBLY    |
|                 | RESI_GLOB_RELA | RESI_GLOB_MAXI |                  |       RHO       |                 |
-------------------------------------------------------------------------------------------------------------
|      0        X | 1.81843E-04  X | 4.64155E-01    |      0           | - WITHOUT OBJECT - |TANGENT        |
|      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 the moment 229.725 , one cut out twice the basic list and one converges in 8 iterations of Newton. One note:

- Linear research was not very expensive: an iteration only (by default STAT_NON_LINE/RECH_LINEAIRE/ ITER_LINE_MAXI=3). It is also seen that there is no research linear in prediction.
- Convergence is tested on the criterion RESI_GLOB_RELA . Without changing the values by default of the order STAT_NON_LINE/ CONVERGENCE , 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 steps of time, one sees that it is always calculated with the first iteration. This corresponds well to the adjustment by default of the order STAT_NON_LINE/ NEWTON: REAC_INCR=1 and REAC_ITER = 0.

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

```
    Statistics on all the transient.
      * Many steps of time                      : 358
      * Iteration count of Newton               : 4353
      * Many factorizations of the matrix      : 358
      * Many integrations of the behavior       : 8715
      * Many resolutions K.U=F                 : 4353

      * Iteration count of linear research   : 4004

    Time CPU spent in the transient          : 2 m 29 S
of which time "wasted" in cuttings   : 55,640 S - > the list of moment 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 law of behavior, 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 step of time (one also sees it on the number of factorizations: 358, like the number of steps of time).
- The initial list of time cut out in 50 pas de time was not most effective: the time wasted in calculation because of failures of convergence and thus of the redécoupe of the step of time is of approximately a third of total time.

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

```
Statistics on all the transient.
          * Many steps of time                      : 50
          * Iteration count of Newton               : 152
          * Many factorizations of the matrix      : 152
          * Many integrations of the behavior       : 202
          * Many resolutions K.U=F                 : 152


        Time 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 the speed of convergence (30 times faster), one observes no cutting of the step of time. One can even more coarsely cut out the list of moments.

5.3.4 Calculation elastoplastic in load then discharge

We now will carry out the discharge. For that, 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 $t=300.$.

Then a news should be defined the list of moments in report: Order `DEFI_LIST_REEL` (30 pas de time until $t=230.$, then 10 pas de time until $t=300.$) and of the order `DEFI_LIST_INST` by activating the automatic cutting of the step of time with `FAILURE/EVENEMENT='ERROR' and ACTION='CUTTING')`.

Do a new calculation (oneE new order `STAT_NON_LINE`) while taking `INST_FIN = 300.` in the keyword `INCREMENT` of the order `STAT_NON_LINE` , and in utilisant the new slope and the new list of moment.

If one takes again the strategy allowing to carry out calculation until the end in plasticity (i.e. until $p=230\,MPa$ while using `STAT_NON_LINE/NEWTON/REAC_ITER=1`), 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 deformation when the loading is null. So that calculation converges, it is thus necessary to activate the elastic matrix in prediction (`STAT_NON_LINE/ NEWTON/PREDICTION=' ELASTIQUE'`).

It is a question of concluding elastoplastic calculation with isotropic work hardening given by a traction diagram such as the uniaxial constraint tends towards a constant value ($270\,MPa$).

There thus exists a limiting load for this structure of which a terminal supérieure is known: $p_{\lim}<243\,MPa$. In this modeling, one will show how to carry out calculation beyond this load limits thanks to piloting.

5.4 Sizes tested and results

One tests the value of the components of constraints for the elastic design (loading of $10\mathrm{MPa}$), one must find the same thing that in modeling A, that is to say:

Component	Type of reference	Value	Tolerance
`SIGM_NOEU` – $SIYY$ in B	`AUTRE_ASTER`	Identical A	

SIGM_NOEU – $SIXX$ in A	AUTRE_ASTER	Identical A

One tests the value of the components of constraints and the internal variables for elastoplastic calculation (at the moment corresponds to the loading of 230MPa):

Component	Type of reference	Tolerance
SIGM_NOEU – $SIYY$ in B	NON_REGRESSION	1,00E-006%
SIGM_NOEU – $SIXX$ in A	NON_REGRESSION	1,00E-006%
VARI _NOEU – $V1$ in B	NON_REGRESSION	1,00E-006%
VARI _NOEU – $V2$ in B	NON_REGRESSION	1,00E-006%
VARI _NOEU – $V1$ in A	NON_REGRESSION	1,00E-006%
VARI _NOEU – $V2$ in A	NON_REGRESSION	1,00E-006%

One tests the value of the components of constraints and the internal variables for elastoplastic calculation with discharge (at the moment corresponds to the final unloading of 0):

Component	Type of reference	Tolerance
SIGM_NOEU – $SIYY$ in B	NON_REGRESSION	1,00E-006%
SIGM_NOEU – $SIXX$ in A	NON_REGRESSION	1,00E-006%
VARI _NOEU – $V1$ in B	NON_REGRESSION	1,00E-006%
VARI _NOEU – $V2$ in B	NON_REGRESSION	1,00E-006%
VARI _NOEU – $V1$ in A	NON_REGRESSION	1,00E-006%
VARI _NOEU – $V2$ in A	NON_REGRESSION	1,00E-006%

6 Modeling C

6.1 Characteristics of modeling

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

- Elastoplastic calculation: one charges until $p=243\,MPa$;
- Elastoplastic calculation: one does a calculation by piloting beyond the limiting load;

6.2 Characteristics of the grid

One uses the same grid as the modeling B which comprises 315 TRIA6 and 686 nodes.

6.3 Calculation with limiting load

6.3.1 Detection " manual " limiting load

To launch the module AsterStudy .
Then in left column, to click on the mitre View box .
One defines the command file of the calculation case (to click on the right with CurrentCase and to choose Add Internship).
Foot-note: to add orders by Menu Commands → All show .

The command file is almost identical with modeling the preceding one. The only difference is that we propose here to activate the automatic management of the list of moments, i.e. the temporal discretization is entirely managed by STAT_NON_LINE .

- To see the grid with format MED: Order LIRE_MAILLAGE.
- To direct Lhas normal edge on which the loading of traction will be applied : Category Mesh / Order MODI_MAILLAGE/ORIE_PEAU_2D in affecting the group hauT in GROUP_MA. One keeps the same name of the grid while using reuse.
- To define the finite elements used: Order AFFE_MODELE for to affect the phenomenon MECHANICS and modeling in plane constraints 2D (C_PLAN) with all the elements .
- To see the traction diagram provided in the file forma03c.21 Order LIRE_FONCTION.
- To define material: Order DEFI_MATERIAU/ ELAS and TRACTION.
- To affect material: Order AFFE_MATERIAU.
- To affect conditions with limiting kinematics : Order AFFE_CHAR_MOVIES / MECA_IMPO for Symetry on the quarter of plate (groups left and low).
- To affect the loading : Order AFFE_CHAR_ MECA/ FORCE_CONTOUR for the force distributed on the top of the plate. Simplest is to define a unit stress (FY=1.0) , that one will multiply then by a function crawls with time.
- To create a function crawls linear $f=t$ for to multiply the mechanical loading unit : Order DEFI_FONCTION. For example, it variE enter $(0.,0.)$ and $(1000.,1000.)$
- **To activate the automatic management of the step of time** with the order DEFI_LIST_INST/METHODE='WithUTO' and DEFI_LIST with PAS_MINI=1.e-6, PAS_MAXI=100 and VALE= (0. , 50. , 243) who give the three moments of obligatory passage of the automatic list.
- Calculer evolution ofelastoplastic: Commande STAT_NON_LINE / BEHAVIOR/RELATION=' VMIS_ISOT_TRAC' with the list of moment defined previously.

One initially proposes to note "with the hand" the limiting load. The limiting load corresponds to the moment when a point of Gauss reaches the equivalent value of Von Mises of approximately 270MPa . By analytical calculation (see §2.2), a limit was determined supérieure of the loading which causes a plasticization with this value limits 270MPa (around the hole).
For that, to proceed as in modeling the preceding one but made a loading beyond $p=230\text{MPa}$. Initially, a value of $p=245\text{MPa}$ is a good reference.

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

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

6.3.2 Calculation beyond the load limits by piloting

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 piloting of the constraint 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 constraint σ_{yy} imposed on FG . One will increase it until $6\,mm$ for example (in preceding calculation, one observed that following maximum displacement Y point G was of approximately 5.7mm , 6mm is thus well beyond the limiting load).

One will take a coefficient equal to 1. A fictitious time will thus be used t such as $\Delta t=\Delta U_Y(G)\times 1$. I.e. time fiction vary here enters 0 and $6\,s$ (to represent a displacement of piloting `DY` G enter 0 and $6\,mm$).

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

To launch the module `AsterStudy` .
Then in left column, to click on the mitre `View box` .
One defines the command file of the calculation case (to click on the right with `CurrentCase` and to choose `Add Internship`)
Foot-note: to add orders by Menu `Commands` → `All show` .

Principal stages for the creation and the launching of the calculation case are the following ones:

- To see the grid with format MED: Order `LIRE_MAILLAGE` .
- To direct Lhas normal edge on which the loading of traction will be applied : Category `Mesh` / Order `MODI_MAILLAGE/ORIE_PEAU_2D` in affecting the group `hauT` in `GROUP_MA`. One keeps the same name of the grid while using `reuse`.
- To define the finite elements used: Order `AFFE_MODELE` for to affect the phenomenon `MECHANICS` and modeling in plane constraints 2D (`C_PLAN`) with all the elements .
- To see the traction diagram provided in the file forma03c.21 Order `LIRE_FONCTION` .
- To define material: Order `DEFI_MATERIAU/ ELAS` and `TRACTION`.
- To affect material with all the elements : Order `AFFE_MATERIAU`.
- To affect conditions with limiting kinematics : Order `AFFE_CHAR_MOVIES` / `MECA_IMPO` for Symetry on the quarter of plate (groups `left` and `low`).

- **To affect the loading** : Order AFFE_CHAR_ MECA/ FORCE_CONTOUR for the force distributed on the top of the plate. One is defined unit stress (FY=1.0) who will be controlled in STAT_NON_LINE.
- **Create the temporal discretization** usingS orders DEFI_LIST_REEL (10 pas de time until $t=4s$) and DEFI_LIST_INST by activating the automatic cutting of the step of time: FAILURE/EVENEMENT='ERROR' and ACTION='CUTTING') .
- **ModifyR the type of the loading (force distributed)** : in the order STAT_NON_LINE / EXCIT, utiliser TYPE_CHARGE='FIXE_PILO' for the force applied. And hasctiver piloting: keyword factor PILOTING :
  - TYPE=' DDL_IMPO' .
  - COEF_MULT=1.0 .
  - GROUP_NO=' G' .
  - NOM_CMP=' DY' .
- Calculationer evolution ofelastoplastic: Order STAT_NON_LINE / BEHAVIOR/RELATION= ' VMIS_ISOT_TRA C '.

ObserveZ in the file "message" the value of the parameter ETA_PILOTAGE, η . To know the load exit of piloting, it is enough to make $F_{pilote}=\eta\times F_y$ with $F_y=1.0$, unit stress which one imposes. One obtains in theory a good approximation of the limiting load (by higher value) than one will be able to compare on the terminal supanalytical érieure ($243\,MPa$).

One will be able to plot, in continuation, the curve forces resultant-displacement in G according to time.

Extraction of the parameter of piloting:

- To extract the values from the parameter ETA_PILOTAGE in the result: order RECU_FONCTION/ NOM_PARA_RESU on the parameter ETA_PILOTAGE .
- To print the function $\mathrm{ETA\_PILOTAGE}=f(t)$: Order IMPR_FONCTION .

One can visualize with Salomé, the deformation, the isovaleurs of constraints $SIYY$ and of the cumulated equivalent plastic deformation. At moment 6, one can notice on the isovaleurs of cumulated plastic deformation, the localization of the deformations in the vicinity of B .

6.4 Sizes tested and results

One tests the value of the limiting load for elastoplastic calculation without piloting, compared to the analytical solution of the minimal terminal:

Component	Type of reference	Value	Tolerance
SIGM_NOEU – $SIYY$ in G	ANALYTICAL	$243\,MPa$	1,00%

One tests the value of the limiting load for elastoplastic calculation with piloting (vertical displacement imposed of $6\,mm$ at the point G), compared to the analytical solution of the minimal terminal, the solution without piloting:

Component	Type of reference	Value	Tolerance
SIGM_NOEU – $SIYY$ in G	ANALYTICAL	$243\,MPa$	1,00%
SIGM_NOEU – $SIYY$ in G	AUTRE_ASTER	$243.05\,MPa$	0.30%
ETA_PILOTAGE in INST=6	ANALYTICAL	$243\,MPa$	1,00%

7 Modeling D

7.1 Characteristics of modeling

Identical to modeling C, this modeling uses a following loading, SUIV_PILO instead of FIXE_PILO (type of loading: PRES_REP instead of FORCE_CONTOUR).

If FIXE_PILO , the loading is always fixed (independent of the geometry).
If S UIV_PILO , The loading is known as “follower”, i.e. it depends on the value of the unknown factors: for example, the pressure, being a loading applying in the normal direction to a structure, depends on the geometry brought up to date of this one, and thus on displacements.

7.2 Characteristics of the grid

One uses the same grid as modeling C who comprises 315 TRIA6 and 686 nodes.

7.3 Sizes tested and results

One tests the value of the limiting load for elastoplastic calculation without piloting, compared to the analytical solution of the minimal terminal:

Component	Type of reference	Value	Tolerance
SIGM_NOEU – $SIYY$ in G	ANALYTICAL	$243\,MPa$	1,00%

One tests the value of the limiting load for elastoplastic calculation with piloting (vertical displacement imposed of 6mm at the point G), compared to the analytical solution of the minimal terminal of the solution without piloting:

Component	Type of reference	Value	Tolerance
SIGM_NOEU – $SIYY$ in G	ANALYTICAL	$243\,MPa$	1,00%
SIGM_NOEU – $SIYY$ in G	AUTRE_ASTER	$243.05\,MPa$	0.30%
ETA_PILOTAGE in INST=6	ANALYTICAL	$246\,MPa$	1,00%

8 Summary of the results

This test makes it possible to show how to carry out the calculation of an elastoplastic structure and its examination, and in particular to highlight the benefit to use piloting for a problem of limiting load. One can retain of this test some ideas:

- Even apart from a perfect elastoplastic behavior, it can exist a limiting load. It is the case with all the real traction diagrams. It is then necessary to adapt the method of resolution to the mechanical solution and for example to use piloting;
- Cutting in small increments of load is often necessary to integrate the relation of behavior correctly. That can also help with convergence, it is thus advised to use the automatic recutting of the step of time;
- Linear research can be used to help with convergence, as well as the automatic subdivision of the steps of time. In the event of discharge, the elastic prediction is an effective solution.