
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.

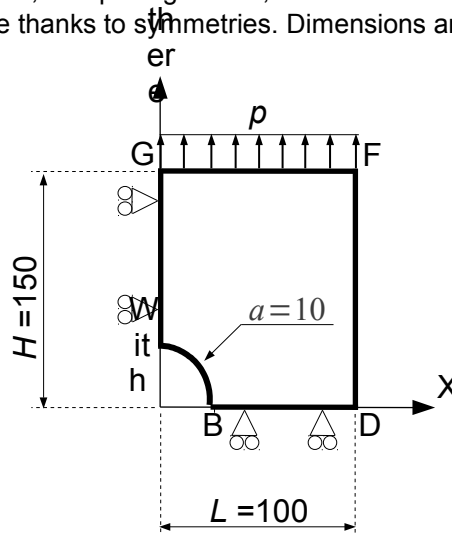
Contents

1	Problem of reference.....	3
1.1	Geometry.....	3
1.2	Boundary conditions and loadings.....	3
1.3	Properties of materials.....	3
2	Reference solution.....	5
2.1	Elastic solution.....	5
2.2	Elastoplastic solution (load limits).....	5
2.3	Bibliographical references.....	6
3	Implementation of the TP.....	7
3.1	Unfolding of the TP.....	7
3.2	Geometry.....	7
3.3	Grid.....	7
3.4	Elastic design with STAT_NON_LINE.....	8
3.4.1	Realisation of calculation.....	8
3.4.2	PostprocessingS results hasvec Aster_Study.....	8
3.5	Elastoplastic calculation with STAT_NON_LINE.....	10
3.5.1	Preparation of the command file.....	10
3.5.2	Calculation rubber band.....	11
3.5.3	Calculation elastoplastic in load.....	11
3.5.4	Calculation elastoplastic in load then discharge.....	13
3.6	Calculation with limiting load.....	14
3.6.1	Detection “ manual “ limiting load.....	14
3.6.2	Calculation beyond the load limits by piloting.....	15
4	Modeling A.....	17
4.1	Characteristics of modeling.....	17
4.2	Characteristics of the grid.....	17
4.3	Sizes tested and results.....	17
5	Modeling B.....	18
5.1	Characteristics of modeling.....	18
5.2	Characteristics of the grid.....	18
5.3	Sizes tested and results.....	18
6	Modeling C.....	19
6.1	Characteristics of modeling.....	19
6.2	Characteristics of the grid.....	19
6.3	Sizes tested and results.....	19
7	Summary of the results.....	20

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 O_x along the side AG and following O_y along the side BD .

Loading in imposed constraint

It is subjected to a traction p according to O_y distributed on the side FG .

1.3 Properties of materials

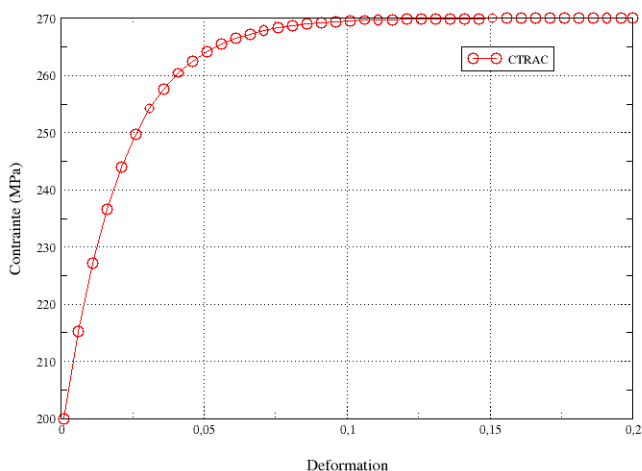
The behavior is elastoplastic of Von Mises, with isotropic work hardening.

Elastic characteristics are:

- Young modulus $E = 200\,000\text{ MPa}$;
- 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'`):

Courbe de traction



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:

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

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

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

In particular, at the edge of the 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 for an infinite plate

Not	Component	MPa
A	SIGXX	- 10
B	SIGYY	30

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 30.3 MPa at the point B .

2.2 Elastoplastic solution (load limits)

In elastoplasticity, by a static approach in plane constraints, one can obtain a lower limit of the limiting load 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 :

$$p_{\text{lim}}^- = \frac{\sigma_y \cdot (L - a)}{L} \quad (6)$$

Here one obtains as limits lower limiting load: $p_{\text{lim}}^- = 0.9 \times 270 = 243 \text{ MPa}$. (One takes here $\sigma_y = 270 \text{ MPa}$, because the limiting load is identical between a perfect elastoplastic material 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 lower limit of it. 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` → `Low ic` → `2D Sketch`). It is simpler of commen cer by not `B` of coordinates `(10,0)` . On the basis of `B` , for the arc of a circle, to use `Direction/Perpendicular` , and to define the angle and the ray. The point is obtained `A` . Then D onner other points 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 the groups D be edges on which will be pressed the boundary conditions (symmetries and loading) (`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. Then, to click on `Add` . A number of object must then appear. One can change the name of the group before `LE` to validate by `Apply` . To build the groups of edge thus following for calculation : `left` for the edge `AG` , `high` for the edge `GF` , `right-hand side` for the edge `FD` , `low` for the edge `DB` and `hole` for the arc `BA` .
- One can also create the groups being based on the tops, they will be useful to create groups of nodes (for postprocessing or piloting) (`Menu New Entity` → `Group` → `Create Group`). To select the geometrical type of entity (here `LE top`, `vertex`) and to select the top directly in the chart window. Then, it is necessary to click on `Add`. A number of object must then appear. One can change the name of the group before `LE` to validate by `Apply` , To build the useful groups of tops thus: five tops `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, then the algorithm `Netgen 1D-2D` with the assumption `Netgen 2D Parameters`. On this assumption, it is enough to select `Fineness` → `Fine Very` and coachman the box `Second Order` . One then obtains a grid refined close to the hole with large elements in the top of the plate.
- To refine this grid, it is necessary to decrease `Max Size` in the parameters of `Netgen`, while choosing for example `10` .

- If one wants to refine around the hole, it is possible to use the cards of size in Netgen:
 - To publish the Netgen parameters.
 - To select the mitre Room sizes.
 - To add the hole group with the button One Edge . It is then enough to modify the associated value. By decreasing it, the grid will be refined on the hole.
- Calculer it grid (Finely Mesh → Compute) .
- To create L be groups D E e-mail be geometrical correspondents with the group (Menu Mesh → Create Groups F Romanian Geometry) . S election ner all geometrical groups .
- Export er it grid with format MED.

3.4 Elastic design with STAT_NON_LINE

It is a question of concluding the elastic design by generating the command file Aster Study using the platform Salomé-Meca . Modeling is C_PLAN . A quarter of the plate is modelled. This modeling makes it possible to carry out a calculation in elasticity for p going until 10 MPa . One must find the same results as by using the order MECA_STATIQUE .

3.4.1 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 (Finely Operations → 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 the grid towards the affected edge by the loading of Neumann : Order MODI_MAILLAGE/ORIE_PEAU_2D while using the group hau T . To put the same name as that of the order LIRE_MAILLAGE .
- To define the finite elements used: Order AFFE_MODELE for modeling in plane constraints 2D (C_PLAN) .
- To define material: Order DEFI_MATERIAU .
- 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.
- 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.
- To create a function crawls linear unit varying enters 0. and 1000. to apply the mechanical loading: Order DEFI_FONCTION .
- To create the temporal discretization usingC MommandeS DEFI_LIST_REEL and DEFI_LIST_INST .
- To calculate the elastic evolution: Order STAT_NON_LINE / BEHAVIOR/ RELATION= ` Elastic (ELAS) ` with the list of moment defined previously. To also put them boundary conditions. If one indicates E Nd time step (INST_FIN) =10 D years the keyword Timestepping (INCREMENT) , one will have applied a force well of 10 MPa .

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

3.4.2 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 Storing (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:

- The field of the constraints with the nodes (SIGM_NOEU). It is necessary to use for that the order CALC_CHAMP/ CONSTRAINT .
- The field of the internal variables to the nodes (VARI_NOEU). It is necessary to use for that the order CALC_CHAMP/ VARI_INTERNE .
- Equivalent constraints (Von Mises, Tresca, etc) at the points of Gauss with the option SIEQ_ELGA of the order CALC_CHAMP/ CRITERIONS .

One proposes then several postprocessings more evolved:

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

To extract the values from displacement at the point G in the result by using the order RECU_FONCTION at once DEPL , component DY .

To extract the values from the constraint at the point G in the result by using the order RECU_FONCTION at once SIGM_NOEU , component SIYY .

To print the function $SIYY=f(DY)$: Order IMPR_FONCTION with the format XMGRACE.

Extraction of the constraint SIYY on the lower edge BD :

The 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 , on the way BD .

To print the table resulting from the order MACR_LIGN_COUPE while filtering over the last moment and by displaying the columns of the curvilinear X-coordinate ABSC_CURV and of the constraint SIYY : Order IMPR_TABLE .

Extraction of the constraints on the edge of the hole:

The order is used MACR_LIGN_COUPE : the constraints are extracted $\sigma_{\theta\theta}$ along the edge of the hole by projecting the components SIXX , SIYY and SIXY DU field SIGM_ELNO in the reference mark POLAR (keyword LIGN_COUPE/REFERENCE MARK) . A table thus is produced .

To extract $\sigma_{\theta\theta}=f(s)$ with s the curvilinear X-coordinate (ABSC_CURV) in the preceding table using the order 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) with the assistance order FORMULA who depends on the curvilinear coordinate s (NOM_PARA='VALE=' p* (1.+2.*cos (2.*S/R))' with p=10MPa and R=10mm).

Interpolation of the formulation by the definition of a list of realities (Order DEFI_LIST_REEL)

dividing the curvilinear coordinate of 0 with $s_{\max}=\frac{\pi}{2.R}$ then use of the order CALC_FONC_INTERP .

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

Calculation of the option `FORC_NODA` with the order `CALC_CHAMP/ FORCE` .

To extract the values from displacement at the point `G` in the result by using the order `RECU_FONCTION` at once `DEPL` , component `DY` .

The order is used `MACR_LIGN_COUPE` : one calculates the resultant of the nodal forces (keyword `RESULTING LIGN_COUPE/`) along the group `high` . A table thus is produced.

To extract $DY=f(t)$ in the preceding table using the order `RECU_FONCTION` .

3.5 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 whose lower limit is known $p_{\text{lim}} > 243\text{ MPa}$. In this modeling, one charges only until 230 MPa and one proceeds to an elastic return. The loading case limits will be treated in LE following paragraph.

3.5.1 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 (`Finely Operations` → `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_MALLAGE` .
- To direct the grid towards the affected edge by the loading of Neumann : Order `MODI_MALLAGE/ORIE_PEAU_2D` while using the group `haut` . To put the same name as that of the order `LIRE_MALLAGE` .
- To define the finite elements used: Order `AFFE_MODELE` for modeling in plane constraints 2D (`C_PLAN`).
- **To see the traction diagram provided in the file `forma03b.21`: Order `LIRE_FONCTION` .**
- **To define material: Order `DEFI_MATERIAU/ Linear isotropic elasticity (ELAS) and TRACTION` .**
- To affect material: 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.
- To affect conditions with limits mechanics and 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`.
- **To create a first function `crawls` linear unit varying enters `0` . and `1000` . to apply the mechanical loading: Order `DEFI_FONCTION` .**
- **To create the temporal discretization using `S` orders `DEFI_LIST_REEL (30 pas de time)` and `DEFI_LIST_INST` by activating the automatic cutting of the step of time: `FAILURE/EVENEMENT= ' ERROR ' and ACTION= ' CUTTING ')` .**
- **To calculate the evolution elastoplastic : Order `STAT_NON_LINE (BEHAVIOR/ RELATION= ' VMIS_ISOT_TRAC ')` with the list of moment defined previously. To also put them boundary conditions.**

3.5.2 Calculation rubber band

If one indicates `END time step (INST_FIN) = 1 0 D` years the keyword `Timestepping (INCREMENT)` order `STAT_NIN_LINE`, one will have applied a force well of 10 MPa , what is equivalent strictly to the elastic case (paragraph § 3.4).

Here some elements to be checked:

- To check that one finds 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 plastic deformation cumulated (`VARI_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 (see “ **Extraction of the constraints on the edge of the hole** “ in the paragraph §3.4.2).

3.5.3 Calculation elastoplastic in load

If one indicates `END time step (INST_FIN) = 23 0 D` years the keyword `Timestepping (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 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`).
- Withctiver 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`).
- Lcombination of all the preceding techniques has.

Here Ucombination which does not function in our case :

- Discretisation of the loading in 50 pas;
- `STAT_NON_LINE/ CONVERGENCE/ITER_GLOB_MAXI = 20` ;
- Activated linear research;
- Dscoop 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 $66,7\text{ MPa}$, 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 ITERATION	RESIDUE RELATIVE RESI_GLOB_RELA	RESIDUE ABSOLUTE RESI_GLOB_MAXI	RESEARCH. LINE. NB. ITER	RESEARCH. LINE. COEFFICIENT RHO	OPTION ASSEMBLY
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	X 8.04731E-07	X 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.

3.5.4 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 End time step (`INST_FIN`) = 300. in the keyword `Timestepping` (`INCREMENT`) of the order `STAT_NON_LINE` , and while not forgetting to use 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 whose lower limit is known: $p_{lim} > 243 MPa$. In this modeling, one will show how to carry out calculation beyond this load limits thanks to piloting.

3.6 Calculation with limiting load

3.6.1 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 (`Finely Operations` → `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_MALLAGE`.
- To direct the grid towards the affected edge by the loading of Neumann : Order `MODI_MALLAGE/ORIE_PEAU_2D` while using the group `haut`. To put the same name as that of the order `LIRE_MALLAGE`.
- To define the finite elements used: Order `AFFE_MODELE` for modeling in plane constraints 2D (`C_PLAN`).
- To see the traction diagram provided in the file `forma03c.21` Order `LIRE_FONCTION`.
- To define material: Order `DEFI_MATERIAU/` Linear isotropic elasticity (`ELAS`) and `TRACTION`.
- To affect material: Order `AFFE_MATERIAU`.
- To affect conditions with limiting kinematics and the loading : Order `AFFE_CHAR_MOVIES / MECA_IMPO` for S ymetry on the quarter of plate.
- To affect conditions with limits mechanics and 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`.
- To create a function `crawls` linear variable enters `0.` and `1000.` to apply the mechanical loading: Order `DEFI_FONCTION`.
- To activate the automatic management of the step of time with the order `DEFI_LIST_INST/METHODE='WithUTO'` and `DEFI_LIST` with `VALE= (0. , 50. , 243)`, `PAS_MINI=1.e-6` and `PAS_MAXI=100`. The list `VALE` give the three moments of obligatory passage of the automatic list.
- Calcular evolution of elastoplastic: 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), one determined a lower limit 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 $p=240\text{MPa}$ 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 $54,44^\circ$ (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 .

3.6.2 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 6mm 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 $\Delta t = \Delta U_Y(G) \times 1$. Thus time varies here enters 0 and 6s (to represent a displacement enters 0 and 6mm).

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 (Finely Operations → 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_MALLAGE`.
- To direct the grid towards the affected edge by the loading of Neumann: Order `MODI_MALLAGE/ORIE_PEAU_2D` while using the group `haut`. To put the same name as that of the order `LIRE_MALLAGE`.
- To define the finite elements used: Order `AFFE_MODELE` for modeling in plane constraints 2D (`C_PLAN`).
- To see the traction diagram provided in the file `forma03c.21` Order `LIRE_FONCTION`.
- To define material: Order `DEFI_MATERIAU/Linear isotropic elasticity (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.
- 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 $t=4$) 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 / Loads (EXCIT)`, utiliser Standard Load (`TYPE_CHARGE`)=`'FIXE_PILO'` for the force applied. And hasctiver piloting: keyword factor `PILOTING`:
`TYPE='Enforce DOF (DDL_IMPO)'`
`COEF_MULT=1.0`
`GROUP_NO=' G'`
`NOM_CMP=' DY'`
- Calculation er evolution of elastoplastic: Order `STAT_NON_LINE / BEHAVIOR/RELATION= ' VMIS_ISOT_TRA C '`.

ObserveZ in the file "message" the value of the parameter `ETA_PILOTAGE`. One obtains in theory a good approximation of the limiting load (by higher value) than one will be able to compare on the analytical lower terminal (243 MPa).

One will be able to plot, in continuation, the curve forces resultant-displacement in G according to time. One should find on this curve the value of the load limits given by '`ETA_PILOTAGE`' .

Extraction of the parameter of piloting:

To extract the values from the coefficient of piloting in the result by using the order `RECU_FONCTION/NOM_PARA_RESU` on the parameter `ETA_PILOTAGE` .

To print the function `ETA_PILOTAGE=f(t)` : **Order** `IMPR_FONCTION` .

One can visualize with Salomé, the deformation, the isovaleurs of constraints S_{IYY} 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 .

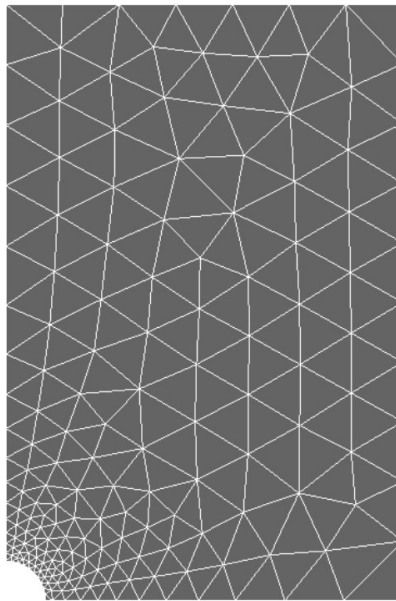
4 Modeling A

4.1 Characteristics of modeling

Elastic design on a model in plane constraints (C_PLAN). The loading is defined in the § 1.2 . One charges until $p=10\text{ MPa}$.

4.2 Characteristics of the grid

A grid is used.



4.3 Sizes tested and results

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

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 $p = 10 \text{ MPa}$;
- Elastoplastic calculation: one charges until $p = 230 \text{ MPa}$;
- Elastoplastic calculation then discharge: one charges until $p = 230 \text{ MPa}$ then one discharges until $p = 0$.

5.2 Characteristics of the grid

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

5.3 Sizes tested and results

One tests the value of the components of constraints for the elastic design (loading of 10 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 230 MPa):

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 $p=243\text{ MPa}$;
- 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 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 and by not-regression:

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, the solution without piloting and by not-regression:

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 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.