# FORMA01 - Travaux practices of the formation “Initiation”: calcul of a plate perforated in rubber band and adaptation of grid

**Summary:**

This test 2D in plane constraints quasi-static allows a catch in hand of the platform Salomé-Meca on a simple case in linear elasticity.

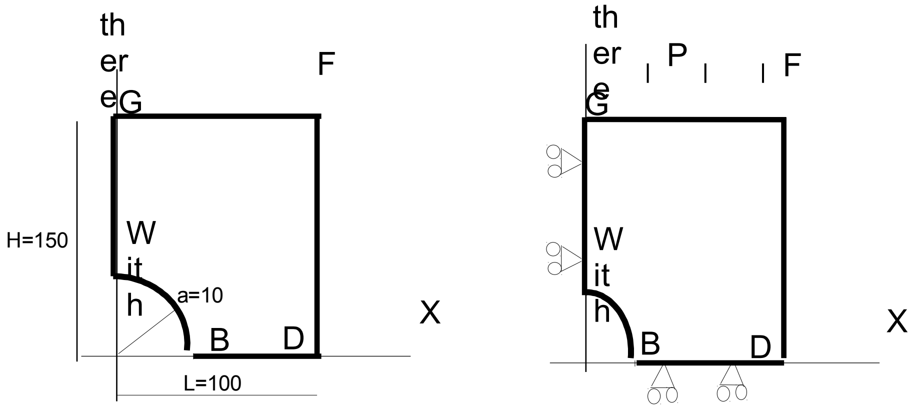
It is about a homogeneous rectangular plate, perforated in its center, which is subjected to a traction at its ends.

Initially, it is indicated how to build the geometry then the grid and how to put in data the study. In the second time, one will carry out the adaptation of the grid.
Lastly, one presents an alternative solution using method X-FEM.

# 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 $mm$ .




## 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 $P=100\,MPa$ according to $Oy$ distributed on the side $FG$ .

## 1.3 Properties of materials

The characteristics are:

- Young modulus $E=200\,000\,MPa$
- Poisson's ratio $\nu=0{,}3$
- Elastic limit: $200\,MPa$

# 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 $(r,\theta)$ is [bib1]:

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

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

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

In particular, at the edge of the hole ( $r=a$ ): $\sigma_{\theta\theta}=P[(1+2\cos 2\theta)]$

And along the axis $x$ : $\sigma_{\theta\theta}=\sigma_{yy}=\frac{P}{2}\left[\left(1+\left(\frac{a}{r}\right)^2\right)+\left(1+3\left(\frac{a}{r}\right)^4\right)\right]$

Numerically, for $P=1\,MPa$, and for a plate **infinite**, one a:

| Not | Component | $MPa$ |
|---|---|---|
| $A$ | $SIXX$ | – 1 |
| $B$ | $SIYY$ | 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 $1\,MPa$, $SIYY$ maximum is worth approximately $3{,}03\,MPa$ at the point $B$.

## 2.2 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. R.E. PETERSON Ed. J. WILEY p150

# 3 Modeling A

## 3.1 Unfolding of the TP

It is a question of concluding the elastic design by generating the geometry and the grid using SalomeMeca, and the assistant and command file Aster using Eficas. Modeling is C_PLAN rubber band. A quarter of the plate is modelled. One will also define the orders necessary to the examination (traced graphic curves and postprocessings).

### 3.1.1 Geometry

One will create the plane face of the quarter higher right it plates it.

The principal stages are after the launching of module GEOM of Salomé-Meca:

| |
|---|
| Creation of contour :<br>One can for example use New Entity/Base/Sketch2D. It is simpler to turn in the direct direction, while starting with the point $B$ for example, of coordinates $(10,0)$ . On the basis of $B$ , for the arc of a circle, to use "Perpendicular", and to define the angle and the ray. Then to give the other points by their absolute coordinates. To finish by "Closure sketch". A closed contour is then obtained (Wire1). |
| Creation of the face: New Entity/Build/Face while basing itself on contour Wire1. |
| Creation of the groups:<br>The groups should be built (" edges ") on which the boundary conditions (symmetries and loading) will rest. For that to select " Create Group " by selecting the object face (click-right), then to select the geometrical type of entity (here the line), to select the edge directly on the graph. Then it is necessary to click on " Add ". A number of object must then appear. One can change the name of the group before validating it by " Apply ". To thus build the 3 useful groups of edge for calculation. |

### 3.1.2 Grid

One will create a grid plan of the quarter higher right of the plate, in elements of order 2, to have a sufficient precision.
The principal stages are after the launching of module SMESH of Salomé-Meca:

| |
|---|
| Creation of the grid :<br>*Create Mesh* , selection of the geometry to be netted, then algorithm of discretization : here *Netgen1D-2D* with the assumption *Netgen2D Parameters* and values *Fine* and *Second order* . |
| Calculation of the grid: *Compute*<br>One then obtains a free grid which comprise approximately 114 TRIA6 and 259 nodes.[1] |
| Creation of the groups of the grid:<br>One creates the geometrical groups of meshs corresponding to the group by *Create Groups From Geometry*, selection of all the geometrical groups: one obtains 3 groups of *edges* on the grid. These groups will be named: $gauche$ , $bas$ , $haut$ . |
| Export of the grid to format MED. |

1 *This grid is generated with version 7.3 of Salomé, it to have little there light differences with other versions.*

**Note:**

*This grid is sufficiently fine (with quadratic elements) to have a good approximation of the solution in elasticity: for example if one compares* $\sigma_{\theta\theta}$ *on the edge of the hole compared to the analytical solution, one obtains a variation of less than 5%.*

*The geometry and the parameters of grid are defined in the file `forma01a.datg` associated with the test. Produced grid is stored in the file `forma01a.mmed`.*

## 3.2 Creation and launching of the calculation case via astk

One opens the module ASTER and using Eficas, one defines the command file of the calculation case.

The principal stages are:

| |
|---|
| Reading of the grid (LIRE_MAILLAGE) with format MED; |
| Definition of the finite elements used (AFFE_MODELE) with a modeling C_PLAN ; |
| Reorientation of the normals to the elements: one will use MODI_MAILLAGE/ORIE_PEAU_2D in the same way to direct all the elements, with a normal turned towards outside (to use the group 'high'); |
| Definition and assignment of material (DEFI_MATERIAU and AFFE_MATERIAU); |
| Definition of the limiting conditions and loadings (AFFE_CHAR_MECA); |
| Symmetry on the quarter of plate (DDL_IMPO); |
| Traction (FORCE_CONTOUR or PRES_REP); |
| Resolution of the elastic problem (MECA_STATIQUE); |
| Calculation of the stress field by elements to the nodes (option 'SIGM_ELNO' of CALC_CHAMP); one will enrich the concept resulting from MECA_STATIQUE in taking again the same name of concept. |
| Calculation of the stress field equivalent by elements to the nodes (option 'SIEQ_ELNO' of CALC_CHAMP/CRITERIA); one will enrich the concept resulting from MECA_STATIQUE in taking again the same name of concept. |
| Calculation of the stress field equivalent by elements to the points of Gauss (option 'SIEQ_ELGA' of CALC_CHAMP); one will enrich the concept resulting from MECA_STATIQUE in taking again the same name of concept. |
| Impression of the results to format MED (IMPR_RESU). |

One creates a calculation case with the command file and the grid. For that one adds in the mitre study of astk a line for the file `*.comm`, `*.mmed`, `*.mess` and `*.resu`. It is also necessary to add some for the file of result `*.rmed`.

One launches the calculation case.

**Note:**

*Although a little more complicated for such a simple calculation, this mode of launching is generic. It gives access all the features of astk and thus to manage complex studies comprising of many data files.*

## 3.3 Creation and launching of the calculation case via the module Aster

One creates a calculation case with the assistant of elasticity. One will take care during the creation of the calculation case of notching the option `Save result database` who allows to have a base containing all the results.

One launches the calculation case.

**Note:**

*The groups defined on the geometry and used in calculation are automatically created on the grid if they are not already present.*

## 3.4 Postprocessing of the results

### 3.4.1 Post-pro: Use of the module of postprocessing

Following postprocessings are proposed:

- To visualize the deformation of the plate (*Deformed Shape*) with the standard of displacement in Iso-values (*Magnitude coloring*) superimposed with the initial grid (*Create representation* on *onCells* then passage of representation *Wireframe*),
- To check starting from the Iso-values of the constraints to the nodes which one finds a relationship 3 between the constraint $\sigma_{\theta\theta}$ at the edge of the hole and the force applied.

### 3.4.2 Paravis: Use of the module of postprocessing

Following postprocessings are proposed:

- To import the file of results (*Open Paraview Slips by*) then Apply
- To visualize the initial grid (passage of representation *Wireframe*)
- To visualize the deformation of the plate (Filter *Warp by Vector*) with the standard of displacement in Iso-values (`Vectors` = *RESU_DEPL_Vector and ScaleFactor = 100* ) superimposed with the initial grid,

To check starting from the Iso-values of the constraints to the nodes which one finds a relationship 3 between the constraint $\sigma_{\theta\theta}$ at the edge of the hole and the force applied.

### 3.4.3 Use of Stanley

Any calculation Aster produces a base which corresponds to the whole of the objects calculated in a study. One uses the possibility of opening the base directly while selecting `Run Stanley` by click-right on this one.

It is by way of a continuation the calculation Aster in which one can make postprocessings.

Postprocessing following is proposed:

- To visualize the field `SIGM_NOEU`,
- To plot the curve of `SIYY` along the segment $BD$ ,
- To visualize the field `SIEQ_ELGA`, field at the point of Gauss.

**Note:**

*The field at the points of Gauss is visualized in the form of spheres located at the points of Gauss of the elements and whose size and color vary according to the value of the field at this place.*

## 3.5 Sizes tested and results

Value of the components of constraints:

| Localization | Identification | Reference (Analytical) | Tolerance |
|---|---|---|---|
| Node $B$ | Constraint $SIYY$ | 303,0 | 5,0% |
| Node $A$ | Constraint $SIXX$ | -100,0 | 15,0% |

These tests on analytical values are doubled by tests of not-regression.

# 4 Modeling B

## 4.1 Unfolding of the TP

It is a question of implementing the adaptation of grid following an elastic design.
One will thus set out again of the study carried out at the time of modeling A.

## 4.2 Adaptation of grid

Using Eficas, one modifies the command file to calculate the indicator of error (option 'ERME_ELEM' of CALC_ERREUR)

One launches calculation and one carries out following postprocessing:

| |
|---|
| To visualize SIYY field SIEF_ELGA, in particular at the edge of the hole, and to record the maximum value about ( $260\,MPa$ ),<br>If one is in PARAVIS, the filter should before be applied Integration Points/Gauss Points and to pass the mode of representation of Surface with Not Sprite. |
| To visualize the cartography of the indicator of error (field ERME_ELEM, component ERREST, total error) |

One modifies the command file to carry out an adaptation of grid and to then directly calculate the new value of indicator of error:

| |
|---|
| To insert the order MACR_ADAP_MAIL for an adaptation of the type 'REFINEMENT'on the result with the field of adaptation'ERME_ELEM'and the component'ERREST' with a criterion of 10% of the elements to have a significant effect, (one will name the new grid MAIL2). |
| To connect the orders then AFFE_MODELE, AFFE_MATERIAU, AFFE_CHAR_MECA, MECA_STATIQUE and CALC_CHAMP on this new grid. |

One launches calculation and one carries out following postprocessing:

| |
|---|
| See the effect of the adaptation on the maximum of SIYY stress field. The maximum is now about $280\,MPa$ . |
| To visualize the cartography of the indicator of error (component ERREST, total error). |

## 4.3 Sizes tested and results

Value of the components of constraints for the initial grid and after an adaptation:

| Grid | Localization | Identification | Reference (Analytical) | Tolerance |
|---|---|---|---|---|
| 1 | Node $B$ | Constraint $SIYY$ | 303,0 | 5,0% |
| 1 | Node $A$ | Constraint $SIXX$ | -100,0 | 15,0% |
| 2 | Node $B$ | Constraint $SIYY$ | 303,0 | 2,0% |
| 2 | Node $A$ | Constraint $SIXX$ | -100,0 | 3,0% |

These tests on analytical values are doubled by tests of not-regression.

# 5 Modeling C

## 5.1 Unfolding of the TP

It is about to implement an adaptation of grid in a loop python on an elastic design.

## 5.2 Adaptation of grid

Starting from the command file of modeling A, one adds a loop python to carry out two adaptations. One will be able to refer to the file `forma01c.comm`.

One launches calculation and one carries out following postprocessing:

| |
|---|
| See the effect of the adaptation on the maximum of `SIYY` stress field. The maximum is now about $280\,MPa$ . |
| To visualize the cartography of the indicator of error (component `ERREST`, total error) |

## 5.3 Sizes tested and results

Value of the components of constraints for the initial grid and after an adaptation:

| Grid | Localization | Identification | Reference (Analytical) | Tolerance |
|---|---|---|---|---|
| 1 | Node $B$ | Constraint $SIYY$ | 303,0 | 5,0% |
| 1 | Node $A$ | Constraint $SIXX$ | -100,0 | 15,0% |
| 2 | Node $B$ | Constraint $SIYY$ | 303,0 | 2,0% |
| 2 | Node $A$ | Constraint $SIXX$ | -100,0 | 3,0% |
| 3 | Node $B$ | Constraint $SIYY$ | 303,0 | 1,0% |
| 3 | Node $A$ | Constraint $SIXX$ | -100,0 | 2,0% |

These tests on analytical values are doubled by tests of not-regression.

# 6 Modeling D

## 6.1 Unfolding of the TP

It is a question of carrying out elastic simulation without netting the hole, by using method X-FEM.
For that, one uses the concept of interface (delimiting the hole), an interface being seen like one "infinite" crack (not of face of crack. The interface is then represented by only one Level Set (Level Set Normal), separating two solids, without contact on the interface. One models thus a hole (vacuum) because the solid inside is insulated moreover structure to which the loading of pressure is applied.

One will generate the geometry and the grid using SalomeMeca, and the command file Aster using Eficas. Modeling is `C_PLAN` rubber band. A quarter of the plate is modelled.

### 6.1.1 Geometry

The geometry is the quarter higher right of the rectangular plate, without hole.

The principal stages are after the launching of module GEOM of Salomé-Meca:

| |
|---|
| Creation of the face:<br>One can for example use `Right-angled New Entity/Primitive/`, then possibly to relocate. |
| Creation of the groups:<br>The groups should be built (" `edges` ") on which the boundary conditions (symmetries and loading) will rest. For that to select " `Create Group` " by selecting the object face (click-right), then to select the geometrical type of entity (here the line), to select the edge directly on the graph. Then it is necessary to click on " `Add` ". A number of object must then appear. One can change the name of the group before validating it by " `Apply` ". To thus build the 3 useful groups of edge for calculation (bottom, the left edge and top). |

### 6.1.2 Grid

One will create a grid plan of the quarter higher right of the plate, in elements of order 2, to have a sufficient precision.
The principal stages are after the launching of module SMESH of Salomé-Meca:

| |
|---|
| Creation of the grid :<br>*Create Mesh* , selection of the geometry to be netted, then algorithm of discretization : here *Netgen1D-2D* with the assumption *Netgen2D Parameters* and values *Fine* and *Second order* . |
| Calculation of the grid: *Compute* |
| Creation of the groups of the grid:<br>One creates the geometrical groups of meshs corresponding to the group by *Create Groups From Geometry*, selection of all the geometrical groups: one obtains 3 groups of *edges* on the grid. These groups will be named: *gauche* , *bas* , *haut* . |
| Export of the grid to format MED. |

## 6.2 Creation and launching of the calculation case (*via* astk)

One opens the module ASTER and using Eficas, one defines the command file of the calculation case.

The principal stages are:

| |
|---|
| Reading of the grid (`LIRE_MAILLAGE`) with format MED; |

| |
|---|
| Reorientation of the normals to the elements: one will use MODI_MAILLAGE/ORIE_PEAU_2D in the same way to direct all the elements, with a normal turned towards outside (to use the group 'high'); |
| Definition of the classical finite elements (AFFE_MODELE) with a modeling C_PLAN ; |
| Definition of the interface (DEFI_FISS_XFEM); |
| Creation of the finite elements nouveau riches (MODI_MODELE_XFEM): model X-FEM |
| Definition and assignment of material (DEFI_MATERIAU and AFFE_MATERIAU) on model X-FEM; |
| Definition of the limiting conditions and loadings (AFFE_CHAR_MECA) on model X-FEM:<br>- Symmetry on the quarter of plate (DDL_IMPO) on model X-FEM;<br>- Traction (FORCE_CONTOUR or PRES_REP); |
| Resolution of the elastic problem (MECA_STATIQUE); |
| Creation of the grid and result of visualization (POST_MAIL_XFEM, POST_CHAM_XFEM); |
| Calculation of the stress field equivalent by elements to the nodes and the points of Gauss (options 'SIEQ_ELNO' and 'SIEQ_ELGA' of CALC_CHAMP/CRITERIA); one will enrich the concept resulting from POST_CHAM_XFEM in taking again the same name of concept. |
| Calculation of the stress field by elements to the nodes (option 'SIGM_ELNO' of CALC_CHAMP); one will enrich the concept resulting from POST_CHAM_XFEM in taking again the same name of concept. |
| Impression of the results to format MED (IMPR_RESU). |

**Note:**

*The rigid modes of the hole are blocked by the conditions of symmetry applied. In the case of a modeling of the whole plate, it would have been necessary to block the rigid modes of the plate and also of those of the hole.*

## 6.3 Postprocessing of the results

One will visualize the deformation of the plate then the constraints at the edge of the hole while comparing oneself with the reference solution. One will be able to possibly add an automatic stage of refinement of grid to improve the precision of the results.

**Note:**

*The rigid modes of the hole are blocked by the conditions of symmetry applied. In the case of a modeling of the whole plate, it would have been necessary to block the rigid modes of the plate and also of those of the hole.*

## 6.4 Sizes tested and results

Value of the components of constraints at points A and b:

For that, the field is calculated SIGM_NOEU. Attention, concept of SIGM_NOEU is particular for X-FEM because the elements generated for visualization (by POST_CHAM_XFEM) have double nodes not connect. One thus has several values for a position of node. In this case, there are two nodes located at point A (with dimensions plates) and two nodes located at the point B (with dimensions plates). One thus tests the min and the max for each point.

| Localization | Identification | Reference (Analytical) | Tolerance |
|---|---|---|---|
| Node $A$ | Constraint $SIXX$ | -100,0 | 5,0% |
| Node $B$ | Constraint $SIYY$ | 303,0 | 5,0% |

These tests on analytical values are doubled by tests of not-regression.

# 7 Summary of the results

This test makes it possible to show how to carry out the calculation of an elastic structure and its examination, and in particular to highlight the benefit to use the adaptation of grid to improve the precision of the results.