

FORMA01 - Travaux pratiques de la formation “Initiation” : modeling of a plate perforated in élastilinear city 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.

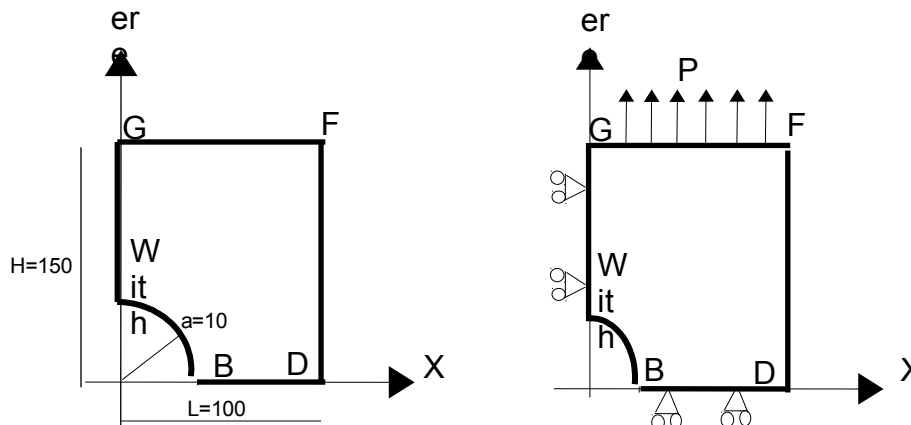
Initially, it is indicated how to build the geometry then the grid and how to put in dataS the study. In the second time, one will carry out the adaptation of the grid.

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. The 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 $P=100 \text{ MPa}$ according to O_y distributed on the side FG .

1.3 Properties of materials

The characteristics are:

- Young modulus $E = 200\,000 \text{ MPa}$
- 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, θ) 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 $P = 1 \text{ MPa}$, and for a plate **infinite**, one a:

Not	Component	MPa
<i>A</i>	<i>S₁₁</i>	-1
<i>B</i>	<i>S₂₂</i>	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 , *S₂₂* maximum is worth approximately $3,03 \text{ MPa}$ 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, the grid and the command file AsterStudy using the platform Salomé-Meca.

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 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 commencer 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 L E to validate by Apply . To thus build the 3 groups of edge useful for calculation : left for the edge AG , high for the edge GF and low for the edge BD .

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.

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, to select Fineness → Fine and coachman the box Second Order .
- Calculer it grid (Finely Mesh → Compute) . A free grid then is obtained.
- To create Lbe groups D E e-mail be correspondents with the group S geometrical (Menu Mesh → Create Groups F Romanian Geometry) . S election ner all geometrical groups. One obtains 3 groups of the edges on the grid.
- Export er it grid with format MED.

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. The produced grid is stored in the file `forma01a.mmed`.

3.2 Creation and launching of the calculation case (via AsterStudy)

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.

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 define L be finite elements used: Order `AFFE_MODELE` for modeling in plane constraints 2D (`C_PLAN`).
- To direct the grid towards the affected edge by the loading: Order `MODI_MALLAGE/ORIE_PEAU_2D` while using the group `haut`.
- To define material: Order `DEFI_MATERIAU`.
- To affect material: Order `AFFE_MATERIAU`.
- To affect conditions with limits mechanics and the loading : `AFFE_CHAR_MECA` :
 - for Symetry on the quarter of plate: Enforce `DOF (DDL_IMPO)`,
 - for Traction: `FORCE_CONTOUR`.
- To solve it problem linear static mechanics : Order `MECA_STATIQUE`.
- To calculate the field : Order `CALC_CHAMP`.

ON will enrich the concept resulting from `MECA_STATIQUE` in taking again the same name of concept.

- for Calcul of the stress field by elements with nodes : `CONSTRAINT/ SIGM_ELNO`.
- for Calcul of the stress field equivalent by elements to nodes : `CRITERIA / SIEQ_ELNO`.
- for Calcul of the stress field equivalent by elements to the points of Gauss: `CRITERIA / SIEQ_ELGA`.
- To print Lbe results of a calculation with format MED: Order `IMPR_RESU`.
- For launchR the calculation case, in left column, to click on the mitre `History View`.

3.3 Postprocessing of the results

To launch the module `ParaVis`.

Following postprocessings are proposed:

- To import the file of results (Mitre `F datedislands` → Open in `ParaVis frontEC` the option `GenerateVectors`).
- To visualize the initial grid (not er in representation `Wireframe`).
- Visualiser deformation of the plate (Menu `Filters` → `Common` → `Warp By Vector` with the options `Vectors = RESU_DEPL_Vector` and `ScaleFactor = 100`) with the standard of superimposed displacement \vec{E} with the initial grid.

To check starting from Lhas value of the constraints to the node B that one finds a relationship 3 between the constraint $\sigma_{\theta\theta}$ at the edge of the hole and the force applied.

3.4 Sizes tested and results

Value of the components of constraints:

Localization	Identification	Reference (Analytical)	Tolerance
Node <i>B</i>	Constraint <i>SIYY</i>	303,0	5,0%
Node <i>A</i>	Constraint <i>SIXX</i>	-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

After having launched the module `AsterStudy`, one modify the command file to calculate the indicator of error:

- Commande `CALC_ERREUR` with the option `ERME_ELEM`.
- To print Lbe results of a calculation with format MED: Order `IMPR_RESU`.
- To launch calculation.

After having launched the module `ParaViS`, one carry out following postprocessing:

- Visualiser the component `SIYY` field `SIEF_ELGA`, in particular at the edge of the hole (Finely Filters → Mechanics → ELGA field To Point Sprite and noter in representation Not Sprite). And Rto raise the maximum value about (260 MPa).
- To visualize the cartography of the indicator of error (field `ERME_ELEM`, component `ERREST`, total error).

After having launched the module `AsterStudy`, one modify the command file to carry out an adaptation of grid and to then directly calculate the new value of indicator of error:

- Commande `MACR_ADAP_MAIL` for an adaptation of the type `RREFINEMENT` on the result with the field of adaptation `ERME_ELEM` and component (`NOM_CMP/ ERREST`) with a criterion of 10% of the elements to have a significant effect (one will name the new grid `MAIL2`).
- To connect then orders `AFFE_MODELE`, `AFFE_MATERIAU`, `AFFE_CHAR_MECA`, `MECA_STATIQUE`, `CALC_CHAMP` and `CALC_ERREUR` on this new grid.
- To print L be results of a calculation with format MED: Order `IMPR_RESU`.
- To launch calculation.

After having launched the module `ParaViS`, one carry out following postprocessing:

- See the effect of the adaptation on the maximum of component `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 <i>B</i>	Constraint <i>SIYY</i>	303,0	5,0%
1	Node <i>A</i>	Constraint <i>SIXX</i>	-100,0	15,0%
2	Node <i>B</i>	Constraint <i>SIYY</i>	303,0	2,0%
2	Node <i>A</i>	Constraint <i>SIXX</i>	-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

After having launched the module `AsterStudthere` and with to leave the command file of modeling A, one adds a loop python to carry out two adaptations (`Finely Operations` → `Text mode`). One will be able to take as a starting point the file `forma01c.comm`.
Calculation is launched.

After having launched the module `ParaViS`, one carry out following postprocessing:

- To voir the effect of the adaptation on the maximum of the component `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 <i>B</i>	Constraint <i>SIYY</i>	303,0	5,0%
1	Node <i>A</i>	Constraint <i>SIXX</i>	-100,0	15,0%
2	Node <i>B</i>	Constraint <i>SIYY</i>	303,0	2,0%
2	Node <i>A</i>	Constraint <i>SIXX</i>	-100,0	3,0%
3	Node <i>B</i>	Constraint <i>SIYY</i>	303,0	1,0%
3	Node <i>A</i>	Constraint <i>SIXX</i>	-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 an "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 to obstruct will era geometry and grid and the command file AsterStudy using the platform Salomé-Meca.

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.

To launch the module `Geometry`.

Principal stages to build this geometry are the following ones :

- To create the face, one can, for example, to use the "right-angled" tool (Menu `Right-angled New Entity/Primitive/`), then possibly to relocate it (`Finely Operations/Transformation/Translation`).
- 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 thus build the 3 groups of edge useful for calculation : `left` for the edge `AG` , `high` for the edge `GF` and `low` for the edge `BD` .

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.

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, to select `Fineness → Fine` and coachman the box `Second Order` .
- Calculer it grid (`Finely Mesh → Compute`).
- To create Lbe groups DE e-mailbe correspondents with the groupS geometrical (Menu `Mesh → Create Groups From Geometry`). Selectionner all geometrical groups. 3 groups D are obtained'edges on the grid.
- Exporter it grid with format `MED`.

6.2 Creation and launching of the calculation case (via AsterStudy)

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.

Foot-note: To add orders: 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: Order `MODI_MALLAGE/ORIE_PEAU_2D` while using the group `haut`.
- To define the finite elements used: Order `AFFE_MODELE` for modeling in plane constraints 2D (`C_PLAN`).
- To define the interface: `DEFI_FISS_XFEM` with the shape of crack (`FORM_FISS/ ELLIPSE`) and the type of discontinuity.
- To create L be finite elements nouveau riches (`MODI_MODELE_XFEM`). And to name new name of model X-FEM.
- To define material: Order `DEFI_MATERIAU`.
- To affect material: Order `AFFE_MATERIAU` on model X-FEM.
- To affect conditions with limits mechanics and the loading : `AFFE_CHAR_MECA` on model X-FEM :
 - for Symetry on the quarter of plate: `Enforce DOF (DDL_IMPO)`,
 - for Traction: `FORCE_CONTOUR`.
- To solve it problem linear static mechanics : Order `MECA_STATIQUE` on model X-FEM.
- To create grid: Order `POST_MAIL_XFEM`. And to name a new name of the grid.
- To define the finite elements used: Order `AFFE_MODELE` for modeling in plane constraints 2D (`C_PLAN`). `E T` to name a new name of the model.
- To create result of visualization: Order `POST_CHAM_XFEM`. And to name a new name of result.
- To calculate the field : Order `CALC_CHAMP`.
 - `ON` will enrich the concept resulting from `POST_CHAM_XFEM` in taking again the same name of concept.
 - for Calcul of the stress field by elements with nodes : `CONSTRAINT/ SIGM_ELNO`.
 - for Calcul of the stress field equivalent by elements to nodes : `CRITERIA / SIEQ_ELNO`.
 - for Calcul of the stress field equivalent by elements to the points of Gauss: `CRITERIA / SIEQ_ELGA`.
 - To print Lbe results of a calculation with format MED: Order `IMPR_RESU`.
 - For launchR the calculation case, in left column, to click on the mitre `History View`.

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

To launch the module `ParaVis`.

One will visualize the deformation of the plate then the constraints at the edge of the hole while comparing oneself with the reference solution.

- To import the file of results (`Mitre F datedislands` → Open in `ParaVis frontEC` the option `GenerateVectors`).
- Visualiser deformation of the plate (Menu `Filters` → `Common` → `Warp By Vector`).

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 connectés. One thus has several values for a position of node. In this case, there are two nodes localisedS at point A (with dimensions plates) and two nodes localisedS at the point B (with dimensions plates). One thus tests the min and the max for each point.

Localization	Identification	Reference (Analytical)	Tolerance
Node <i>A</i>	Constraint <i>SIXX</i>	-100,0	5,0%
Node <i>B</i>	Constraint <i>SIYY</i>	303,0	5,0%

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

7 Summary of the results

This test watch how to carry out the calculation of an elastic structure and how to extract in postprocessing the sizes from interest. It met also in obviousness the benefit to use the adaptation of grid to improve the precision of the results.