Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Date : 13/09/2019 Page : 1/13
Clé : V3.02.326 Révision :
4ba933d27e36

FORMA01 - Travaux practices of the 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 (modeling A), it is indicated how to build the geometry then the grid and how to put in dataS the study. In the second time (modeling B and C), one will carry out the adaptation of the grid. Finally (modeling D), one presents an alternative solution using method X-FEM.

The statements of Practical works of formation (TP1) are included in this document (see modelings A, B and C).

Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Date : 13/09/2019 *Page : 2/13*
Clé : V3.02.326 *Révision :*
4ba933d27e36

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.
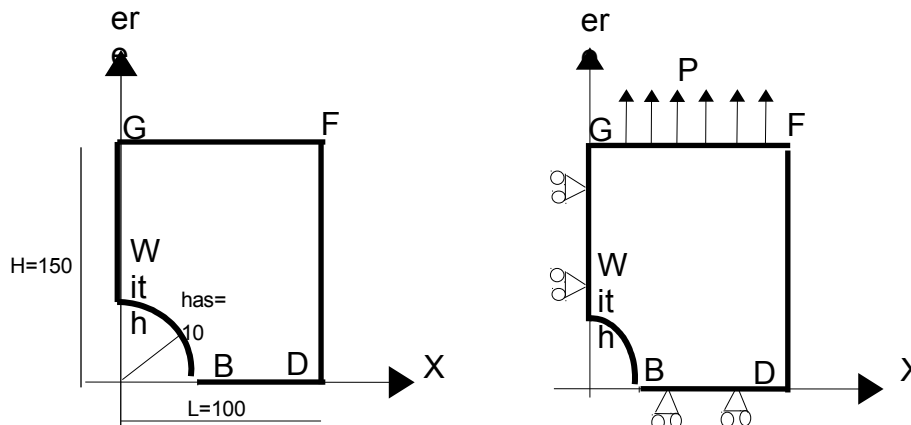
1.2 Boundary conditions and loadings

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 $P = 100\,MPa$ 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: $200\,MPa$

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Version
default

Date : 13/09/2019 Page : 3/13
Clé : V3.02.326   Révision   :
4ba933d27e36

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

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 $1\,MPa$, $SIYY$ 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

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Version default

Date : 13/09/2019 Page : 4/13
Clé : V3.02.326 Révision :
4ba933d27e36

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 → 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 the 3 groups Dbe 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 . 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, Ensuite to click on `Add`, UN number of object must then appear. One can change the name of the group before LE to validate by `Apply`.

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 `Face_1`, then to choose `Withlgorithm → NETGEN 1D-2D` while adding `Hypothelocated → 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 a free grid then.
- To create Lbe groups DE e-mailbe correspondents with the groupS geometrical (Menu `Mesh → Create Groups FRomanian Geometry`) . Selectionner all them 3 groups edges under module `Geometry` and then `Apply`. One obtains 3 groups of the edges on the grid.
- To obtain a better precision, one will make pass the grid of linear to quadratic, thanks to the tool " `Modification - > Convert to/from quadratic` ". Most curious can compare the differences in results between the two types of elements.
- (Optional) Exporter it grid with format MED: to select `Mesh_1` and to click right-hand side, then to choose `Export/MED slips by`.

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

**Version
default**

Date : 13/09/2019 *Page : 5/13*
Clé : V3.02.326 *Révision :*
4ba933d27e36

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. In the box Settings dated, to click on the right with CurrentCase and to choose Add Internship. It is in Stage_1 where ON defines the command file of the calculation case.

Foot-note: to seek orders by Menu Commands → All show however by the 10 categories of calculation: Mesh, Model Definition, Material, Functions and Lists, BC and Load, Pre Analysis, Post Processing, Fracture and Tiredness, Output.

Principal stages for the creation and the launching of the calculation case are the following ones:
- To see the grid with format MED: Category Mesh / Order LIRE_MAILLAGE. Grid in the module MESH is available in the list. If not the button behind makes it possible to choose a grid in the repertoires.
- 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 Lbe finite elements used: Category Model Definition / Order AFFE_MODELE for modeling in plane constraints 2D (C_PLAN). To add a term in AFFE. In this term, to affect YES to ALL elements, and to choose the phenomenon MECHANICS, and to add the term of plane constraints 2D (C_PLAN) in modeling
- To define material: Category Material / Order DEFI_MATERIAU . To choose ELAS and to seize the values of the Young modulus and the Poisson's ratio.
- To affect material: Category Material/ Order AFFE_MATERIAU. To define at least an option enters GRID and MODEL. To add a term in AFFE, and to affect material defined in ALL.
- To affect conditions with limits mechanics and the loading : Category BC and Load / Order AFFE_CHAR_MECA :
  ○ for Symetry on the quarter of plate: DDL_IMPO. For left, DX=0, and for low, DY=0.
  ○ for Traction: PRES_REP. For high, to seize traction with NEAR (thus it is negative).
- To solve it problem linear static mechanics : Category Analysis / Order MECA_STATIQUE by giving material (CHAM_MATER), modeling (MODEL), boundary conditions and the loading (EXCIT).
- To calculate the field : Category Post Processing / Order CALC_CHAMP while activating reuse (for to keep the same name of the object of entry in the order). I.e. ON will enrich the concept resulting from MECA_STATIQUE in taking again the same name.
  ○ for Calcul of the stress field: CONSTRAINT/ SIGM_ELNO (constraints by elements with nodes), SIGM_NOEU (constraints total with the nodes).
  ○ for Calcul of the equivalent stress field : CRITERIA / SIEQ_ELNO (by elements with nodes), SIEQ_ELGA (by elements at the points of Gauss), SIEQ_NOEU (total with the nodes).
- To print Lbe results of a calculation with format MED: Category Output / Order IMPR_RESU . To add a term in RESU, and to choose the results to print. To define the output file (.rmed) by UNIT.
- For launchR the calculation case, in left column, to click on the mitre History View . It is necessary to save the study initially, and to select CurrentCase. To add Stage_1 while selecting run (cross in green). Before launching, checking parameters of calculation desired in the box Run Parameters: time limits of calculation, memory, version of code_aster etc.

Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Date : 13/09/2019 Page : 6/13
Clé : V3.02.326 Révision :
4ba933d27e36

3.3 Postprocessing of the results

To display the results, two choices are currently available: mitre `Results` in AsterStudy, and the module `ParaViS`.

Choice 1 : In mitre `Results` in AsterStudy :
Following postprocessings are proposed:
- To import the file of results (Box `View box → F dated island S` , to click right on the output file and to choose `Post-process` . By default, the colouring of the plate by the field of displacement `DEPL` , an automatic amplification is applied to the deformation of the plate. To visualize well the difference between the initial form and the deformation, to click right to the window and to choose `Show ace/Wireframe` .
- D ouble to click the desired field, to use the button `Honest been worth one one gold more points gold concealments` (symbol cross) to check the values at points A and B.
- (optional) to go to the Paravis module, one finds the actual position of visualization and also all the historical actions.

Choice 2 : In the module `ParaViS` :
Following postprocessings are proposed:
- To import the file of results (Box `View box → F datedislandS`, to click right on the output file and to choose `Open in ParaVis`.
- To download the results: In the mitre `Properties` the column on the left, to initially choose the desired fields to import, and Cocher the option `GenerateVectors`, then hasppliquer `Apply`.
- To visualize the initial grid (noter in representation `Surface With Edges`).
- Visualiser deformation of the plate (Menu `Filters → Common → Warp By Vector` with the options `Vectors = results_DEPL_Vector` and `ScaleFactor = 100` front `Apply`).
- To visualize the stress field by elements with the nodes or the points Gauss: to select the imported result, then in Menu `Filters → Mechanics`, to choose the option of desired visualization.

To check Lhas value of the constraints to the node B: if one finds a relationship 3 between the constraint $\sigma_{\theta\theta}$ (= σ_{yy}) at the edge of the hole and the force applied. To display the results on a node, to activate `Hover points one` (like the photograph below), and to move the mouse with the desired node.

3.4 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

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Version
default

Date : 13/09/2019 Page : 7/13
Clé : V3.02.326 Révision :
4ba933d27e36

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 for to add the calculation of the indicator of error:
- To add results of error with Commande `CALC_ERREUR` in the category `Post Processing` : to choose itoption `ERME_ELEM` while using `reuse`.
- Relancer calculation.

To visualize the error and stress fields (component `ERREST`, total error), one can carry out postprocessing in the mitre `Results` (strongly recommended) or in the module `ParaVis`. Stages in the module `ParaViS` are like according to:
- C ocher fields to visualize to charge them before applying `GenerateVectors` : `SIEF_ELGA` , `ERME_ELEM`
- For visualiser the component `SIYY` field `SIEF_ELGA` in particular at the edge of the hole: Menu `Filters` → `Mechanics` → `ELGA field To Surface` and noter in representation `Surface With EdgeS`. For Rto raise the position of the maximum value about ($270\,MPa$): Menu `Edict` → `Find Dated` and to choose the option `Concealments` in the field creates `ELGAfieldToSurface`, to select `SIEF_ELGA_Vector (1)` representative `SIYY` and formulates it `max. is`.
- To visualize the cartography of the indicator of error (field `ERME_ELEM` , component `ERREST` , total error).

One turns over to 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:
- Category `Post Processing` / Commande `MACR_ADAP_MAIL` for an adaptation of grid of type `RREFINEMENT`: the new grid is named by `MAILLAGE_NP1=MAIL2`, to select the criterion of the elements (option `CRIT_RAFF_PE`) to 10% for the parameter of adaptation: `RESULTAT_IN` / `NOM_CHAM=ERME_ELEM` / Shas component `NOM_CMP = ERREST`.
- To connect then orders `AFFE_MODELE`, `AFFE_MATERIAU`, `AFFE_CHAR_MECA`, `MECA_STATIQUE`, `CALC_CHAMP` and `CALC_ERREUR` on this new grid `MAIL2` (Duplicate orders by modifying the grid and other concerning terms). **Attendtion** : the names of the concepts of these orders should not be identical to those of front, and they do not exceed 8 characters.
- To print Lbe new results DE calculation with format MED: Order `IMPR_RESU` .
- To launch calculation.

To visualize LES fieldS of constraint and of error, one can carry out postprocessing in the mitre `Results` (strongly recommended) or in the module `ParaVis`.

4.3 Sizes tested and results

Value of the components of constraints (`SIGM_NOEU`) with 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%

Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Date : 13/09/2019 Page : 8/13
Clé : V3.02.326   Révision :
4ba933d27e36

2	Node A	Constraint $SIXX$	-100,0	3,0%

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

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Version
default

Date : 13/09/2019 Page : 9/13
Clé : V3.02.326   Révision   :
4ba933d27e36

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:
- Vto oir 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 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.

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

**Version
default**

Date : 13/09/2019 Page : 10/13
Clé : V3.02.326   Révision   :
4ba933d27e36

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 obstructwill 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 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 .

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 FRomanian Geometry`). Selectionner all geometrical groups. 3 groups D are obtained'edges on the grid.
*   Exporter it grid with format MED.

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Version default

Date : 13/09/2019 *Page : 11/13*
Clé : V3.02.326 *Révision : 4ba933d27e36*

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_MAILLAGE` .
- To direct the grid towards the affected edge by the loading: Order `MODI_MAILLAGE/ORIE_PEAU_2D` while using the group `hau T` .
- 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` .
    O N will enrich the concept resulting from `POST_CHAM_XFEM` in taking again the same name of concept.
  - for C alcul 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.*

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Version
default

Date : 13/09/2019 *Page : 12/13*
Clé : V3.02.326 *Révision :*
4ba933d27e36

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 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.

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : BEIGNION Céline

Version default

Date : 13/09/2019 Page : 13/13
Clé : V3.02.326 Révision :
4ba933d27e36

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.