Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : Josselin DELMAS

Date : 19/04/2013 Page : 1/10
Clé : V3.02.326   Révision : 10935

FORMA01 - Practical works of training "Initiation": computation of a plate perforated out of elastic and mesh adaptation

Summarized:

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

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

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

Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : Josselin DELMAS

Date : 19/04/2013 Page : 2/10
Clé : V3.02.326 Révision : 10935

1 Problem of reference

1.1 Geometry

It acts D `a rectangular plate, comprising a hole, modelled in $2D$ plane stresses. One models only one quarter of the plate thanks to symmetries. Dimensions are given in mm .
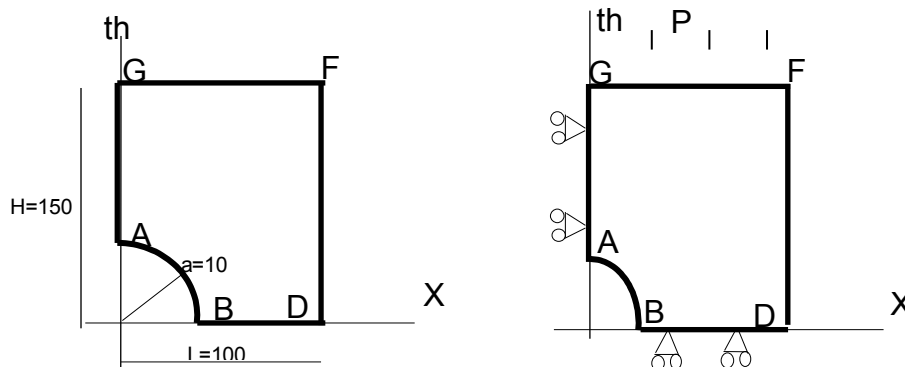
1.2 Boundary conditions and loadings

Conditions of symmetry:

The plate is blocked according to Ox along the side AG and following Oy along the side BD .

Loading in imposed stress:

It is subjected to a tension $P=100$ according to Oy distributed on the side FG .

1.3 Properties of the 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 : Josselin DELMAS

Version
default

Date : 19/04/2013 Page : 3/10
Clé : V3.02.326   Révision : 10935

2   Elastic reference solution

2.1   Solution

In elasticity, for an infinite **plate,** comprising a hole of diameter a, subjected to a loading P according to y ad infinitum, the analytical solution in plane stresses 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 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 an infinite **plate**, one a:

Not	Component	MPa
A	$SIXX$	-1
B	$SIYY$	3

For a plate of finished **size**, the abacuses [bib1] make it possible to obtain the coefficient of stress concentration, and one finds that for a tension of $1\,MPa$, $SIYY$ maximum is worth approximately $3,03\,MPa$ at the point B .

2.2   Bibliographical references

[1]   Limit analysis of fissured structures and strength criteria. F. VOLDOIRE: Note EDF/DER/HI/74/95/26 1995

[2]   Stresses concentration factors. R.E. PETERSON ED. J. WILEY p150

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.

Licensed under the terms of the GNU FDL (http://www.gnu.org/copyleft/fdl.html)

Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : Josselin DELMAS

Date : 19/04/2013 Page : 4/10
Clé : V3.02.326   Révision : 10935

3 Modelization A

3.1 Unfolding of the TP

It acts to conclude the elastic design by generating the geometry and the mesh using SalomeMeca, and the file of Aster commands using Eficas and the assistant. The modelization is `C_PLAN` elastic. A quarter of the plate is modelled. One will also define the commands 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 main steps are after the launching of modulus GEOM of Salome-Meca:

Creation of contour :   One can for example use `New Entity/Base/Sketch2D`. It is simpler to turn in the direct meaning, 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 radius. 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 Wire1 `contour`.
Creation of the groups: It is necessary to build the groups (" edges") on which will rest the boundary conditions (symmetries and loading). For that to select " `Create Group` " by selecting the object face (click-right), then to select the geometrical type of entity (here line), to select 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 computation.

3.1.2 Mesh

One will create a plane mesh of the quarter higher right it plates it, in elements of order 2, to have a sufficient accuracy.
The main steps are after the launching of modulus SMESH of Salome-Meca:

Creation of mesh: *Create Mesh*, selection of the geometry to be netted, then algorithm of discretization : here *Netgen1D-2D* with the assumption *Netgen2D Parameters* and the values *Fine* and *Second order*.
Computation of mesh: *Compute*   One then obtains a free mesh which comprises approximately 113 TRIA6 and 250 nodes.
Creation of the groups of mesh: One creates the mesh groups corresponding to the group geometrical by *Create Groups From Geometry*, selection of all the geometrical groups: one obtains 3 groups of *edges* on the mesh. These groups will be named: *gauche bas haut*.
Export of the mesh to med format.

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : Josselin DELMAS

Version default

Date : 19/04/2013 Page : 5/10
Clé : V3.02.326   Révision : 10935

Note:

> *This mesh is sufficiently fine (with quadratic elements) to have a good approximation of the solution in elasticity: for example if one compared to the $\sigma_{\theta\theta}$ formula on edge of analytical hole solution, one obtains a variation of less than 5%.*
>
> *The geometry and the parameters of mesh are defined in the file `forma01a.datg` `associated` with the test. The mesh product is stored in the file formed `01a.mmed`.* `Creation`

3.2   and launching of the calculation case via astk One opens

modulus ASTER and using Eficas, one defines the command file of the calculation case. The main steps

are: Reading

of mesh (LIRE_MAILLAGE) `to med format`; Definition
of the finite elements used (AFFE_MODELE) `with` a modelization C_PLAN ; `Reorientation`
of the norms to the elements: one will use MODI_ `GRID ORIE_PEAU_2D` in the same way to direct all the elements, with a turned norm towards outside (to use the "high" `group`); `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); `Tension`
(FORCE_CONTOUR `or` PRES_REP); `Resolution`
of the elastic problem (MECA_STATIQUE); `Computation`
of the stress field by elements to the nodes (option "SIGM_ELNO " `of` CALC_CHAMP); `one` will enrich the concept resulting from MECA_STATIQUE `by taking again` the same name of concept. Computation
of the stress field equivalent by elements to the nodes (option "SIEQ_ELNO " `of` CALC _ CHAMP/CRITERES); `one` will enrich the concept resulting from MECA_STATIQUE `by taking again` the same name of concept. Computation
of the stress field equivalent by elements to Gauss points (option "SIEQ_ELGA " `of` CALC_CHAMP); `one` will enrich the concept resulting from MECA_STATIQUE `by taking again` the same name of concept. Printing of results
with med format (IMPR_RESU) . `One` creates

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

the calculation case. Note:

Although

> *a little more complicated for such a simple computation, 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. Creation*

Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]   *Date : 19/04/2013 Page : 6/10*
Responsable : Josselin DELMAS   *Clé : V3.02.326   Révision : 10935*

3.3 and launching of the calculation case via the modulus 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 which makes it possible* to have a base containing all the results. One launches

the calculation case. Note:

The groups
defined on the geometry and used in computation are automatically created on the mesh if they are not already present. Postprocessing

3.4 of the Use results

3.4.1 of the modulus of postprocessing Post-Pro One proposes

following postprocessings: To visualize

- the deformed shape of the plate (Deformed *Shape) with* the norm of displacement in Iso-values (Magnitude *coloring) superimposed* to the initial mesh (Create *representation on* onCells *then* transition of Wireframe representation *), To check*
- from the Iso-values of the nodal stresses which one finds a relationship 3 between the stress at the edge of $\sigma_{\theta\theta}$ hole and the applied force. Use

3.4.2 of the modulus of postprocessing Paravis One proposes

following postprocessings: To import

- the results file (Open Paraview *Slips by) then* Apply To visualize
- the initial mesh (transition of Wireframe representation *) To visualize*
- the deformed shape of the plate (Filter Warp *by Vector) with* the norm of displacement in Iso-values (Vectors = RESU _ *DEPL_Vector and ScaleFactor = 100) superimposed* to the initial mesh, To check

from the Iso-values of the nodal stresses which one finds a relationship 3 between the stress formulates $\sigma_{\theta\theta}$ hole and the applied force. Use

3.4.3 of Stanley Any

Aster computation produces a base which corresponds to all the objects calculated in a study. One uses the possibility of opening the base directly by selecting Run Stanley *by click* - right on this one. It

is by way of a poursuite the Aster computation in which one can make postprocessings.

Postprocessing following is proposed: To visualize

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

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

3.5 tested and Value results

Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : Josselin DELMAS

Date : 19/04/2013 Page : 7/10
Clé : V3.02.326   Révision : 10935

of the components of stresses: Localization

Identification	Reference	(Analytical) Tolerance	Node
Forced B	303,0 $SIYY$	303.0	5.0%
formulates A	formula $SIXX$	-100.0	15.0%

on analytical values are doubled by tests of NON-regression. Modelization

Code_Aster

Version default

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : Josselin DELMAS

Date : 19/04/2013 Page : 8/10
Clé : V3.02.326   Révision : 10935

4   B Unfolding

4.1   of the TP It

acts to implement the mesh adaptation following an elastic design. One will thus set out again
of the study carried out at the time of modelization A. Adaptationde maillage

4.2   Using

Eficas, one modifies the command file to compute: the error indicator (option "ERME_ELEM " of
CALC _ ERREUR) One launches

computation and one carries out following postprocessing: To visualize

SIYY of field SIEF_ELGA , in particular at the edge of hole, and to record the maximum value (about formula $260\,MPa$ one
is in PARAVIS, it is necessary before to apply the filter Integration Points/Gauss Points and to pass the mode of representation of Surface at Sprite Point. To visualize

the cartography of the error indicator (field ERME_ELEM , component ERREST , total error) One modifies

the command file to carry out a mesh adaptation and to then directly calculate the new value of error
indicator: To insert

command MACR_ ADAP_MAIL for an adaptation of the type "RAFFINEMENT" on result with the field of adaptation "ERME_ELEM" and component "ERREST" with a criterion of 10% of the elements to have a significant effect, (one will name new mesh MAIL2). To then connect

the command AFFE_MODELEs , AFFE_MATERIAU , AFFE_CHAR_MECA , MECA_STATIQUE and CALC_CHAMP on this new mesh. One launches

computation and one carries out following postprocessing: See

the effect of the adaptation on the maximum of SIYY of the stress field . The maximum is now about formula $280\,MPa$

the cartography of the error indicator (component ERREST , total error). Quantities

4.3   tested and Value results

of the components of stresses for the initial mesh and after an adaptation: Mesh

Localization	Identification	Reference	(Analytical) Tolerance	1 Node
1	Forced B	303,0 $SIYY$	303.0	5.0%
1	Forced A	-100,0 $SIXX$	-100.0	15.0%
2	Forced B	303,0 $SIYY$	303.0	2.0%
2	Forced A	-100,0 $SIXX$	-100.0	3.0%

on analytical values are doubled by tests of NON-regression. Modelization

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : Josselin DELMAS

Version default

Date : 19/04/2013 Page : 9/10
Clé : V3.02.326   Révision : 10935

5   C Unfolding

5.1   of the TP It

acts to implement a mesh adaptation in a loop python on an elastic design. Mesh adaptation

5.2   From

the command file of the modelization A, one adds a loop python to carry out two adaptations. One will be able to refer to the file formed `01c.comm. One launches`

computation and one carries out following postprocessing: See the effect

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

5.3   tested and Value results of the components

of stresses for the initial mesh and after an adaptation: Mesh

Localization	Identification	Reference	(Analytical) Tolerance	1 Node
1	B	303,0 5,0% $SIYY$	303.0	5.0%
1	A	-100,0 15,0% $SIXX$	-100.0	15.0%
2	B	303,0 2,0% $SIYY$	303.0	2.0%
2	A	-100,0 3,0% $SIXX$	-100.0	3.0%
3	B	303,0 1,0% $SIYY$	303.0	1.0%
3	A	-100,0 2,0% $SIXX$	-100.0	2.0%

on analytical values are doubled by tests of NON-regression. Summary

Code_Aster

Titre : FORMA01 - Travaux pratiques de la formation « Init[...]
Responsable : Josselin DELMAS

Version
default

Date : 19/04/2013 Page : 10/10
Clé : V3.02.326   Révision : 10935

6   of the results This test

makes it possible to show how to carry out the computation of an elastic structure and its examination, and in particular to highlight the benefit to use the mesh adaptation to improve the accuracy of the results.