SSNP142 – Modelization of the fracture in the presence of under pressures and sliding of a stopping with the elements of Summarized

joint:

In this test one models the setting in water of a stopping weight. The various modelizations take into account the propagation of under pressures, the fracture or the sliding (friction) of stopping on the level of its foundation. In the whole case the modelization is carried out using the elements of joint (XXX_JOINT or XXX_JOINT_HYME) and of models JOINT_MECA_RUPT and JOINT_MECA_FROT.

Modelization a: Stopping $2D$, JOINT_MECA_RUPT and under imposed pressures (PLAN_JOINT) linear
Modelization b: Stopping $3D$ [1], JOINT_MECA_RUPT and under imposed pressures (3D_JOINT) linear
Modelization C: Stopping $2D$, JOINT_MECA_FROT and under imposed pressures (PLAN_JOINT) linear
Modelization D: Stopping $3D$, JOINT_MECA_FROT and under imposed pressures (3D_JOINT) linear
Modelization E: Stopping $2D$, JOINT_MECA_RUPT, propagation of under pressures (PLAN_JOINT_HYME)
Modelization F: Stopping $3D$ [1], JOINT_MECA_RUPT, propagation of under pressures (3D_JOINT_HYME)
Modelization G: Stopping $2D$, JOINT_MECA_FROT, propagation of under pressures (PLAN_JOINT_HYME)
Modelization H: Stopping $3D$ [1], JOINT_MECA_FROT, propagation of under pressures (3D_JOINT_HYME)
Modelization I: Stopping $2D$, JOINT_MECA_RUPT and under imposed pressures (PLAN_JOINT) Modelization
will quadra J: Stopping $2D$, JOINT_MECA_FROT and under imposed pressures (PLAN_JOINT) will quadra

the results of modelizations with model JOINT_MECA_RUPT (A, B, E, F and I) are validated by comparison with those obtained with the computer code *GEFDYN,* used currently by the Hydraulic Center of Engineering of EDF.

1 $3D$ is obtained by the extrusion of the model $2D$.

1 Problem of reference

1.1 Geometry

One considers a stopping in the shape of trapezoid of great base $5\,m$, small base $1.5\,m$ and height $10\,m$. This last is posed in the center of a rectangular foundation $15\,m$ length and $5\,m$ top (see figure 1.1). For the modelization $3D$ dimensions are identical in the plane (x, y) and the group is extruded of $1\,m$ in the direction z (of $10\,cm$ for the modelization F).

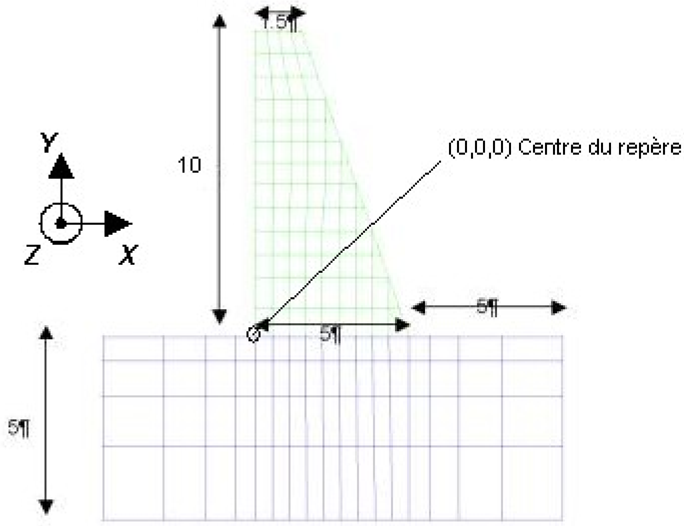

Figure 1.1: Geometry of the stopping and the foundation

1.2 Properties of the material

the values of the mechanical parameters of the stopping (Young modulus, Poisson's ratio, voluminal density of the stopping and water) are in the following way selected:

$$E=3.10^2\,Pa \quad \nu=0.25 \quad \rho_b=2400\,kg/m^3 \quad \rho_e=1000\,kg/m^3$$

1.2.1 Modelizations A, B and I: model `JOINT_MECA_RUPT`

These modelizations test the structural mechanics behavior with under imposed pressures. For the joint one takes the normal stiffness equal to the tangential stiffness. There is no tensile strength. The coupling between the normal opening and the tangential stiffness is selected in order to have the tangential slope null as soon as the joint reaches the threshold of complete normal damage. The slope of softening in fracture is five times stiffer than the normal slope of loading (see document [`R7.01.25`]).

$$K_N=K_T=10^{12}\,Pa/m \qquad \sigma_{max}=0\,Pa$$

$$\alpha=1 \qquad \text{pena\_rupt}=0.2$$

(NB: the values “tests” provided by the CIH do not correspond to any material in particular)

1.2.2 Modelizations C, D and J: model `JOINT_MECA_FROT`

the joint is modelled by an elastoplastic model of standard friction Mohr-Coulomb (see [`R7.01.25`]), which depends on five parameters. Two elastic parameters: one takes the tangential stiffness equal to the double of the normal stiffness. Two parameters of the model Mohr-Coulomb: dependancy and the coefficient of kinetic friction. More one parameter of regularization of the tangent matrix in sliding.

$K_N = 10^{12}\,Pa/m$ $\quad$ $\text{adhésion} = 1\,kPa$ $\quad$ $\text{pena\_tang} = 0.1 \cdot K_T$

$K_T = 2 * K_N$ $\quad$ $\mu = 0.35$

1.2.3 Modelizations E and F: model JOINT_MECA_RUPT

These modelizations test the hydro-mechanical behavior with the propagation of under pressures. For the joint one takes the normal stiffness equal to the tangential stiffness. The tensile strength has a weak value in order to facilitate the convergence of computation. The coupling between the normal opening and the tangential stiffness is selected in order to have the tangential slope null as soon as the joint reaches the threshold of complete normal damage. The slope of softening in fracture is five times stiffer than the normal slope of loading (see document [R7.01.25]). The hydraulic parameters are those of water.

$K_N = K_T = 10^{12}\,Pa/m$ $\quad$ $\sigma_{max} = 100\,Pa$ $\quad$ $\varrho_{\text{eau}} = 1000\,kg/m^3$

$\alpha = 1$ $\quad$ $\text{pena\_rupt} = 0.2$ $\quad$ $visc_{\text{eau}} = 10^{-3}\,Pa \cdot s$

(NB: the values “tests” provided by the CIH do not correspond to any material in particular)

1.2.4 Modelizations G and H: model JOINT_MECA_FROT

These modelizations test the hydro-mechanical behavior with the propagation of under pressures. The joint is modelled by an elastoplastic model of standard friction Mohr-Coulomb (see [R7.01.25]), which depends on five parameters. Two elastic parameters: one takes the tangential stiffness equal to the double of the normal stiffness. Two parameters of the model Mohr-Coulomb: dependancy and the coefficient of kinetic friction. More one parameter of regularization of the tangent matrix in sliding. The hydraulic parameters are those of water.

$K_N = K_T/2 = 10^{12}\,Pa/m$ $\quad$ $\text{adhésion} = 1\,kPa$ $\quad$ $\varrho_{\text{eau}} = 1000\,kg/m^3$

$\text{pena\_tang} = 0.1 \cdot K_T$ $\quad$ $\mu = 0.4$ $\quad$ $visc_{\text{eau}} = 10^{-3}\,Pa \cdot s$

1.3 formula Boundary conditions and

1.3.1 loadings Loadings

the interface between the stopping and the foundation is modelled by of the finite elements joint. The lower parts of the foundation are clamped. The stopping is subjected to the gravitational force and one gradually fills out of water the upstream part of the stopping (left on the figure). That amounts applying a distributed pressure P_{amont} to the face $(x=0, y \in [0,10])$ which one expresses according to y :

$$P_{amont} = \rho_e\, g\,(n_e - y)$$

where g the acceleration of gravity indicates ρ_e the density of water and n_e the water level.

1.3.2 Under pressure

For the pure mechanical modelizations (A, B, C, D, I, J) one imposes a profile of under pressures given.
To model the water propagation in crack under the stopping (in $y=0$) one takes into account a linear fluid pressure P_{fluide} . This one is worth $\rho_e\, g\, n_e$ upstream $(x=0)$ and is null downstream $(x=5)$.
In the modelizations E and F one takes into account the propagation of under pressures (one imposes only the pressure upstream and the pressure downstream), it is the computation which will provide the profile of under pressures.

<u>Note:</u> In a preoccupation with a robustness of computations one uses AFFE_CHAR_CINE for the boundary conditions on the joints (modelizations E and F). Moreover one imposes key word NPREC=-1 in the part solver of STAT_NON_LINE.

2 Reference solution

For the modelizations with model JOINT_MECA_RUPT (A, B, E, F and I), the reference solution is given by the computer code GEFDYN [bib1]. The modelizations C, D, G, H and J are used as test of robustness and non regression of the model of friction.

2.1 Uncertainties on the solution

For GEFDYN, the relative accuracy is fixed at 1E-2 and the mesh is of five to ten times coarser, which explains a variation (and not an error) about 10% between the modelizations GEFDYN and Code_Aster.

2.2 Bibliographical references

[1] GEFDYN, Geomechanical Dynamic Finite elements. Analyzes coupled in Mechanics/Hydraulic/Thermal of the nonlinear behavior of géomatériaux quasi static and/or dynamic 2D/3D. MSSMat laboratory, Central School Paris.

3 Modelization A

3.1 Characteristic of the modelization

simulation is carried out with modelization PLAN_JOINT. The elements are of type TRIA3 for the stopping and the foundation and of type QUAD4 for the elements of joint. The corresponding constitutive law is JOINT_MECA_RUPT, the associated material bears the same name. The surface elements are elastic.

3.2 Characteristics of the mesh

a linear mesh is carried out:
Voluminal elements (stopping and foundation): 2264 TRIA3
Elements of joint: 50 QUAD4

3.3 Quantities tested and results

One compares the results with those of GEFDYN. One notes δ_n (V7) the normal opening of the joints and σ_n (SIGN) the normal stress. The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU).

Quantity tested	*GEFDYN*	*Tolerance*
δ_n in $x=0m$	4.01D-7	7%
δ_n in $x=5m$	-4.25D-7	5%
σ_n in $x=0m$	-8.83D+04	1%
σ_n in $x=5m$	-4.250D+05	5%

Of the tests of non regression are also carried out to make sure of the stability of the data-processing developments, they are not presented here. One represents the deformed shape of the stopping and the opening of crack on figure 3.3 below. To allow a good visualization, a multiplicative factor is applied.

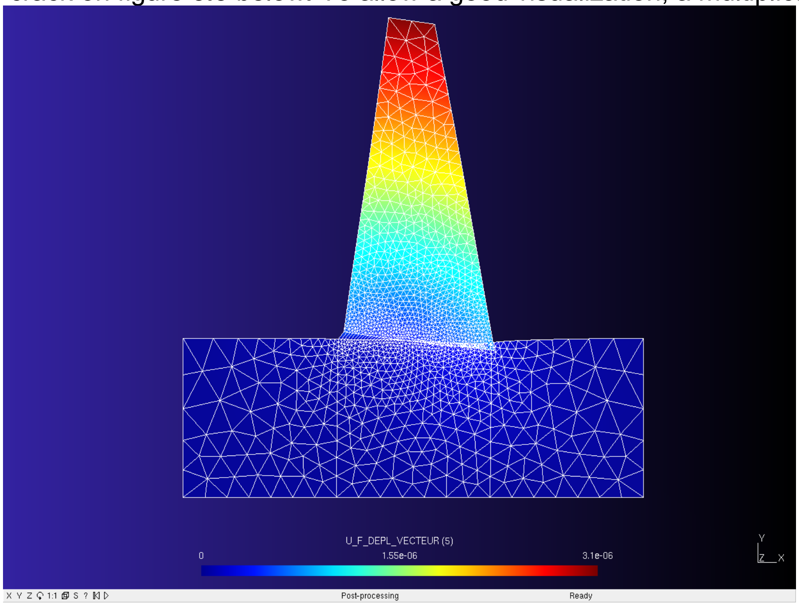

Figure 3.3: Deformed stopping and opening of the crack

4 Modelization B

4.1 Characteristic of the modelization

simulation is carried out with modelization 3D_JOINT. The elements are of type TETRA4 for the stopping and the foundation and of type PENTA6 for the elements of joint. The corresponding constitutive law is JOINT_MECA_RUPT, the associated material bears the same name. The voluminal elements are elastic.

4.2 Characteristics of the mesh

a linear mesh is carried out:
Voluminal elements (stopping and foundation): 37583 TETRA4
Elements of joint: 1146 PENTA6

4.3 Quantities tested and results

One compares the results with those of GEFDYN. One notes δ_n (V7) the normal opening of the joints and σ_n (SIGN) the normal stress. The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU).

Quantity tested	*GEFDYN*	*Tolerance*
δ_n in $x=0\,m$	4.01D-7	5%
δ_n in $x=0\,m$	-4.25D-7	3%
σ_n in $x=5\,m$	-8.83D+04	1%
σ_n in $x=5\,m$	-4.250D+05	3%

Of the tests of non regression are also carried out to make sure of the stability of the data-processing developments, they are not presented here.

5 Modelization C

5.1 Characteristic of the modelization

simulation is carried out with modelization PLAN_JOINT. The mesh is regular. The elements are of type QUAD4 and TIRA3 for the stopping and the foundation and of type QUAD4 for the elements of joint. The corresponding constitutive law is JOINT_MECA_FROT, the associated material bears the same name. The surface elements are elastic.

5.2 Characteristics of the mesh

a linear mesh is carried out:
Voluminal elements (stopping and foundation): 34 TRIA3 and 860 QUAD4
Elements of joint: 50 QUAD4

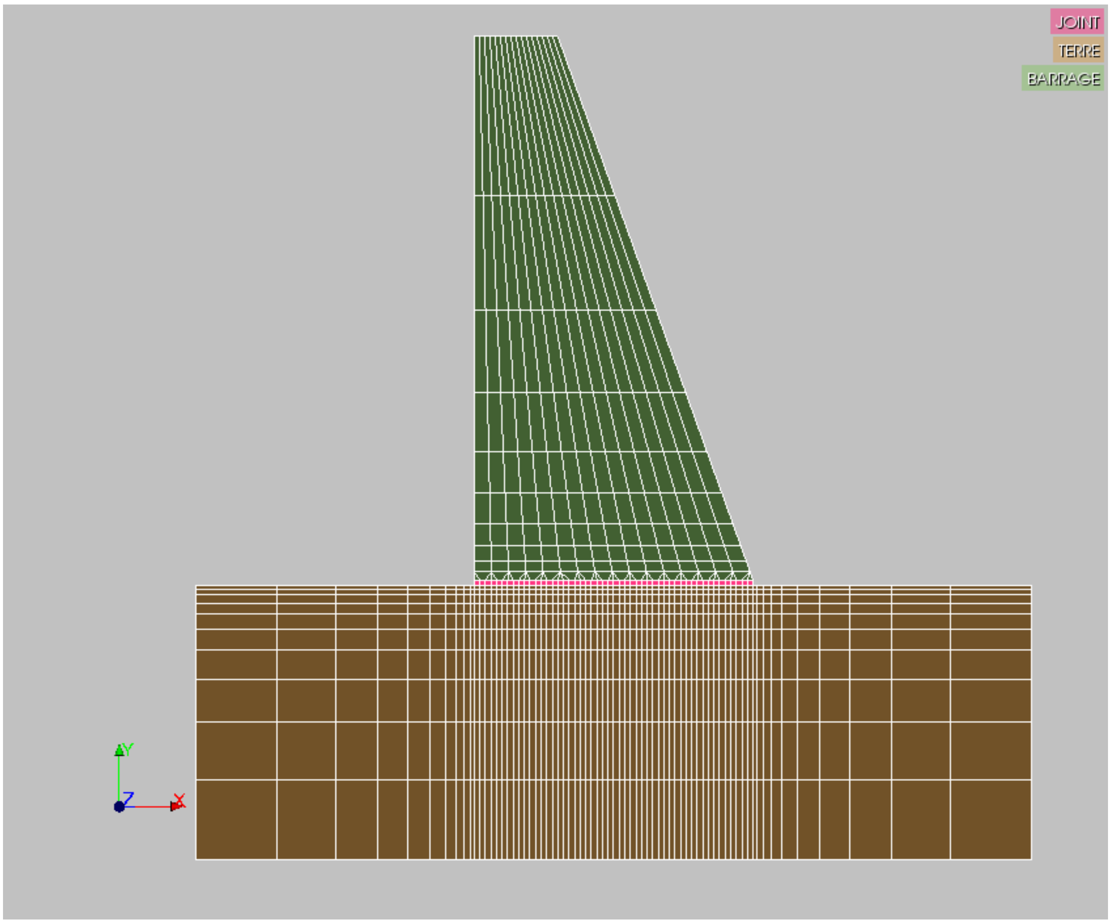

Figure 5.2-1: Regular mesh of the modelization C

5.3 Quantities tested and results

Of the tests of non regression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via V7 or DEPL) the normal opening of the joints, σ_n (SIGN) the normal stress and then the fluid pressure imposed (variable V18). The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU).

6 Modelization D

6.1 Characteristic of the modelization

simulation is carried out with modelization 3D_JOINT. The elements are of type TETRA4 for the stopping and the foundation and of type PENTA6 for the elements of joint. The corresponding constitutive law is JOINT_MECA_FROT, the associated material bears the same name. The voluminal elements are elastic.

6.2 Characteristics of the mesh

a linear mesh is carried out:
Voluminal elements (stopping and foundation): 37583 TETRA4
Elements of joint: 1146 PENTA6

6.3 Quantities tested and results

Of the tests of non regression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via V7 or DEPL) the normal opening of the joints, σ_n (SIGN) the normal stress, then the fluid pressure imposed (variable V18). The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU).

As illustration one presents the visualization of stresses (on the left) and the part in sliding (on the right) of the stopping on figure 6.3.

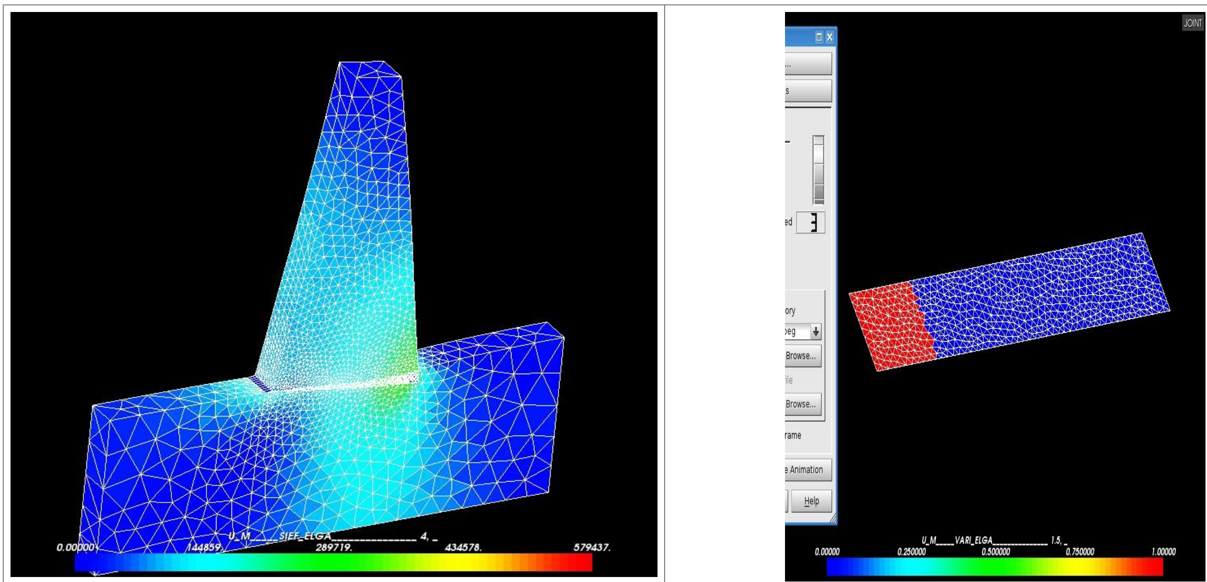

Figure 6.3: Forced on the stopping and the part of the slip plane (in red)

7 Modelization E

7.1 Characteristic of the modelization

simulation is carried out with modelization PLAN_JOINT_HYME. The elements are of type TRIA6 for the stopping and the foundation and of type QUAD8 for the elements of joint. The corresponding constitutive law is JOINT_MECA_RUPT, the associated material bears the same name. The surface elements are elastic.

7.2 Characteristics of the mesh

a quadratic mesh is carried out:
Voluminal elements (stopping and foundation): 3626 TRIA6
Elements of joint: 100 QUAD8

7.3 Quantities tested and results

One compares the results with those of GEFDYN. One notes δ_n (V7) the normal opening of the joints and σ_n (SIGN) the normal stress. The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU).

Quantity tested	***GEFDYN***	***Tolerance***
δ_n in $x=0\,m$	1.42E-06	8%
σ_n in $x=0\,m$	-88290	1%
δ_n in $x=5\,m$	-5.84E-07	15%
σ_n in $x=5\,m$	-5.86E+05	15%

Of the tests of non regression are also carried out to make sure of the stability of the data-processing developments, they are not presented here.

8 Modelization F

8.1 Characteristic of the modelization

simulation is carried out with modelization 3D_JOINT_HYME. The elements are of type TETRA10 for the stopping and the foundation and of type PENTA15 for the elements of joint. The corresponding constitutive law is JOINT_MECA_RUPT, the associated material bears the same name. The voluminal elements are elastic.

8.2 Characteristics of the mesh

a linear mesh is carried out:
Voluminal elements (stopping and foundation): 7435 TETRA10
Elements of joint: 200 PENTA15

8.3 Quantities tested and results

One compares the results with those of GEFDYN. One notes δ_n (V7) the normal opening of the joints and σ_n (SIGN) the normal stress. The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU).

Quantity tested	*GEFDYN*	*Tolerance*
δ_n in $x=0m$	1.44E-06	6%
σ_n in $x=0m$	-88290	1%
δ_n in $x=5m$	-5.89E-07	15%
σ_n in $x=5m$	-5.89E+05	15%

Of the tests of non regression are also carried out to make sure of the stability of the data-processing developments, they are not presented here. One carries out in particular tests on the local variable 18 which makes it possible to know the value of the fluid pressure interpolated with Gauss points.

9 Modelization G

9.1 Characteristic of the modelization

simulation is carried out with modelization PLAN_JOINT_HYME. The elements are of type TRIA6 for the stopping and the foundation and of type QUAD8 for the elements of joint. The corresponding constitutive law is JOINT_MECA_FROT, the associated material bears the same name. The surface elements are elastic.

9.2 Characteristics of the mesh

a quadratic mesh is carried out:
Voluminal elements (stopping and foundation): 3626 TRIA6
Elements of joint: 100 QUAD8

9.3 Quantities tested and results

Of the tests of non regression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via V7 or DEPL) the normal opening of the joints, σ_n (SIGN) the normal stress . The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU). One carries out in particular tests on the local variable 18 which makes it possible to know the value of the fluid pressure interpolated with Gauss points.

10 Modelization H

10.1 Characteristic of the modelization

simulation is carried out with modelization 3D_JOINT_HYME. The elements are of type TETRA10 for the stopping and the foundation and of type PENTA15 for the elements of joint. The corresponding constitutive law is JOINT_MECA_FROT, the associated material bears the same name. The voluminal elements are elastic.

10.2 Characteristics of the mesh

a linear mesh is carried out:
Voluminal elements (stopping and foundation): 7435 TETRA10
Elements of joint: 200 PENTA15

10.3 Quantities tested and results

Of the tests of non regression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via V7 or DEPL) the normal opening of the joints, σ_n (SIGN) the normal stress. The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU). One carries out in particular tests on the local variable 18 which makes it possible to know the value of the fluid pressure interpolated with Gauss points.

11 Modelization I

11.1 Characteristic of the modelization

It is equivalent modelization A into quadratic. Simulation is carried out with modelization PLAN_JOINT. The elements are of type TRIA6 for the stopping and the foundation and of type QUAD8 for the elements of joint. The corresponding constitutive law is JOINT_MECA_RUPT, the associated material bears the same name. The surface elements are elastic.

11.2 Characteristics of the mesh

a quadratic mesh is carried out:
Voluminal elements (stopping and foundation): 2264 TRIA6
Elements of joint: 50 QUAD8

11.3 Quantities tested and results

One compares the results with those of GEFDYN. One notes δ_n (V7) the normal opening of the joints and σ_n (SIGN) the normal stress. The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU).

Quantity tested	***GEFDYN***	***Tolerance***
δ_n in $x=0\,m$	4.01D-7	7%
δ_n in $x=5\,m$	-4.25D-7	5%
σ_n in $x=0\,m$	-8.83D+04	1%
σ_n in $x=5\,m$	-4.250D+05	5%

Of the tests of non regression are also carried out to make sure of the stability of the data-processing developments, they are not presented here.

12 Modelization J

12.1 Characteristic of the modelization

It is equivalent quadratic of the modelization B. simulation is carried out with modelization PLAN_JOINT. The mesh is regular. The elements are of type QUAD8 and TIRA6 for the stopping and the foundation and of type QUAD8 for the elements of joint. The corresponding constitutive law is JOINT_MECA_FROT, the associated material bears the same name. The surface elements are elastic.

12.2 Characteristics of the mesh

a linear mesh is carried out:
Voluminal elements (stopping and foundation): 34 TRIA6 and 860 QUAD8
Elements of joint: 50 QUAD8

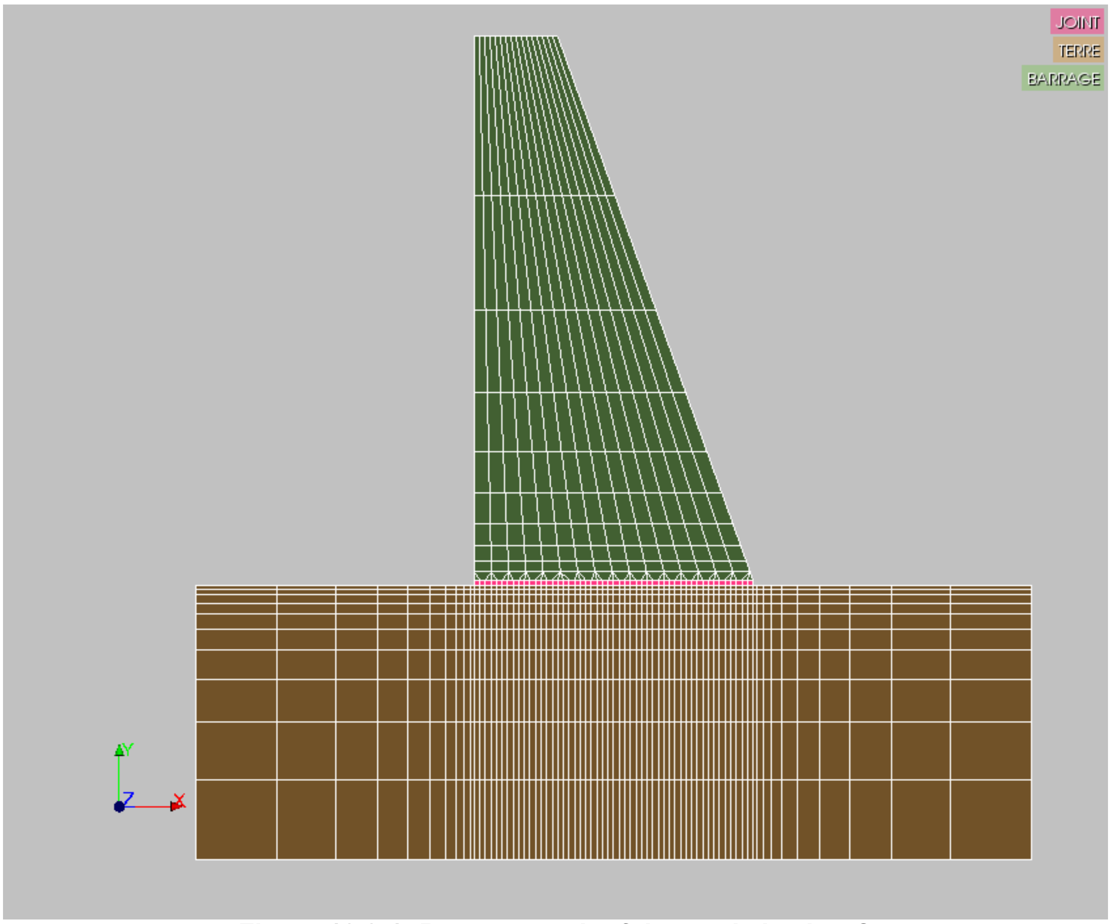

Figure 12.2-1: Regular mesh of the modelization C

12.3 Quantities tested and results

Of the tests of non regression are carried out to make sure of the stability of the data-processing developments. One tests δ_n (obtained via V7 or DEPL) the normal opening of the joints, σ_n (SIGN) the normal stress and then the fluid pressure imposed (variable V18). The values tested result from extrapolations to nodes (VARI_NOEU and SIEF_NOEU).

13 Summary of the results

the modelizations A, B, E, F and I provide results in conformity with GEFDYN. That makes it possible to validate constitutive law JOINT_MECA_RUPT at the same time on a level mechanical and hydro-mechanical, that also validates the taking into account of the fluid pressure on the lips of crack using key word PRES_FLUIDE. The modelizations C, D, G, H and J validate the robustness of the version full-implicit of friction law JOINT_MECA_FROT.