TU Delft
Delft University of Technology
Faculty of Civil Engineering

University of Minho
Department of Civil Engineering

A USER/PROGRAMMER GUIDE FOR THE MICRO-MODELING OF MASONRY STRUCTURES

Author : P. B. LOURENÇO
Date : November 1996

stw

TU-DELFT report no. 03.21.1.31.35
TNO-BOUW report no. 96-NM-R1201

TNO Building and Construction Research
Computational Mechanics

Summary

This report details the micro-model for masonry structures to be available in version 7.1 of the DIANA finite element program. The use of the model is discussed and the new syntax is presented. Comprehensive examples, which include the phases of mesh generation, nonlinear analysis and interpretation of the results, are also given. Finally, all the expressions utilized in the computational code are included for future reference and maintenance.

Acknowledgments

The financial support by the Netherlands Technology Foundation (STW) under grant DCT-33.3052 is gratefully acknowledged.

The calculations have been carried out with the Finite Element Package DIANA of TNO Building and Construction Research on a Silicon Graphics Indigo R4000 workstation of the Delft University of Technology.

1. Introduction

This report presents a comprehensive user's and programmer's guide for the micro-modeling strategy of masonry structures proposed in Lourenço (1996). The strategy consists of modeling units with elastic continuum elements and joints with inelastic interface elements. The composite interface model has been developed according to the plasticity theory.

Chapter 2 presents the additional DIANA syntax needed to use the interface masonry model, including the material data, the proposed automatic arc-length procedures, new layout specifications and an external masonry pre-processor.

Chapter 3 reviews experimental data available for the Dutch materials and provides recommendations for use.

Chapters 4 and 5 present the steps of a nonlinear analysis of two masonry structures, including pre- and post-processing of the results. Chapter 4 includes a masonry pier with a point load and Chapter 5 includes a masonry shear wall with an opening.

Appendix A gives the complete formulation of the interface composite model.

Appendix B includes the source code of the special masonry pre-processor.

2. DIANA additional syntax

This section details the additional DIANA syntax for the use of the interface composite model.

2.1 Material data

In the following, the syntax of the nonlinear material data input for the composite interface model is detailed. The model sets a nonlinear relation between tractions (i.e. stresses) and relative displacements across the interface. The tractions are a normal traction t_n and a shear traction t_t. The relative displacements are a normal displacement Δu_n and a shear displacement Δu_t.

Apart from the input described in this section, the linear stiffness must always be specified as described in Volume *Linear Static Analysis*.

Masonry inelastic behavior may be specified according to the following syntax:

syntax

'MATERI'

1 5 6 12 13 80

MASINT	
GAPVAL	ft_r
MODE1	$mo1_n$
MO1VAL	$mo1_r$
FRCVAL	ch_r tph_r tps_r $[tphu_r$ $tpsu_r$ $unconf_r]$
MODE2	$mo2_n$
MO2VAL	$mo2_r$
CAPVAL	fm_r cs_r
MODEC	moc_n
MOCVAL	$moc1_r$ $moc2_r$

MASINT indicates use of the composite yield surface for masonry, see Figure 1.

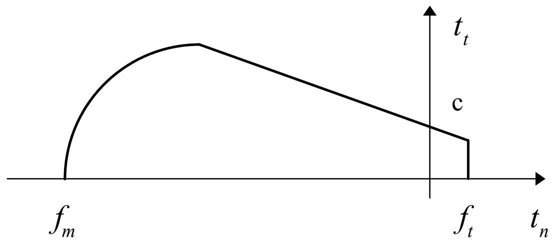


Figure 1 - Composite interface model

GAPVAL *ft* is the tensile strength f_t.

MODE1 *mo*1 indicates the mode I tension softening criterion. Only exponential softening is considered in the current implementation, see Figure 2. *mo*1 = 4

MO1VAL *mo*1 is the fracture energy G_f^I, defining the area under the diagram for mode I.

FRCVAL describes the friction criterion: *ch* is the cohesion c, *tph* is the initial tangent of the friction angle ϕ, *tps* is the initial tangent of the dilatancy angle ψ, *tphu* is the tangent of the residual friction angle ϕ_r, *tpsu* is the tangent of the residual dilatancy angle ψ_r and *unconf* is the value σ_{unconf} of the confining normal stress for which the initial dilatancy angle is zero. $\tan\psi \leq \tan\phi$ $\sigma_{unconf} \leq 0$

MODE2 *mo*2 indicates the mode II shear softening criterion. Only exponential softening is considered in the current implementation, see Figure 3. *mo*2 = 4

MO2VAL *mo*2 is the fracture energy G_f^{II}, defining the area under the diagram for mode II.

CAPVAL describes the cap criterion: *fm* is the compressive strength of masonry f_m and the parameter *cs* controls the contribution of the shear traction to compressive failure via the elliptical cap $\sigma^2 + C_s\,\tau^2 = f_m$.

MODEC *moc* indicates the compression inelastic criterion. Only parabolic hardening followed by parabolic/exponential softening is considered in the current implementation, see Figure 4. *moc* = 5

MOCVAL describes the inelastic law: *moc1* is the compressive fracture energy G_{fc} and *moc2* is the equivalent plastic relative displacement κ_p corresponding to the peak stress.

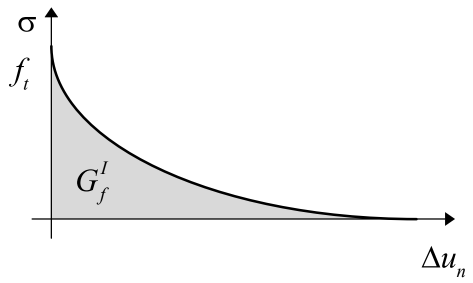


Figure 2 - Behavior of the model in tension (mode I)

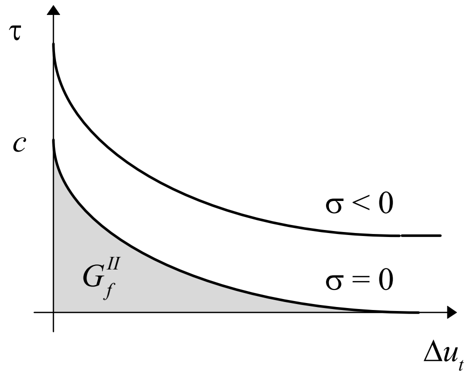


Figure 3 - Behavior of the model in shear (mode II)

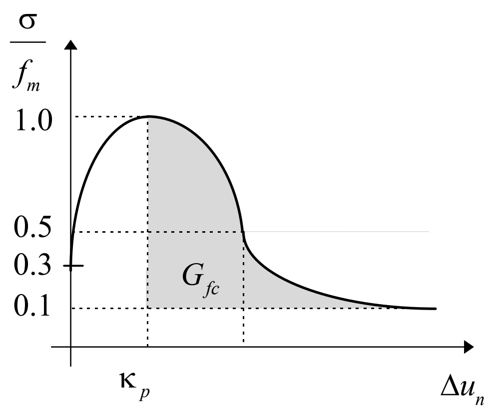


Figure 4 - Behavior of the model in compression

2.2 Indirect control selection SELECT ARCLEN

Two special optional command block of type SELECT ARCLEN has been created: INTTOT and INTREL.

In the first option (INTTOT), a loop over all the nodes connected via an interface element detects the maximum incremental relative displacement (normal or shear). The pair of nodes with the maximum incremental relative displacement is then used with the standard CMOD option. This way, the most critical pair of interface nodes automatically controls the iterative procedure without the need of any input from the user. It is recommended to adopt this new feature throughout the analysis.

In certain extreme cases, for load steps leading to very high energy releases, the second option must be used (INTREL), where the ratio incremental relative displacement / total relative displacement is adopted as control measure.

syntax

```
SELECT ARCLEN INTTOT
END SELECT ARCLEN
```

syntax

```
SELECT ARCLEN INTREL
END SELECT ARCLEN
```

2.3 Layout specification LAYOUT

Additional syntax commands for plotting of analysis results have been created (in bold).

syntax

LAYOUT
 NODES DATA.D**E**VFM
 ELEMEN DATA.VFASLIC123**4567**
END LAYOUT

NODES DATA specifies the plotting style of nodal analysis results.

DATA.E draws only the contour (edges) of the deformed model.

`ELEMEN DATA` specifies the plotting style of element analysis results.

`DATA.F4` or `DATA.F5` plot the data for interface elements with a triangle.

`DATA.F6` or `DATA.F7` plot the data for interface elements with a thick line.

	DATA.F4	DATA.F5	DATA.F6	DATA.F7
positive values	△	△ (dotted)	▬▬▬	········
negative values	△ (dotted)	△	········	▬▬▬

Figure 5 - Additional plotting style of element results as figures for interfaces

2.4 Special masonry pre-processor `MAKE_WALL`

A pre-processor for making regular masonry meshes has been created. This mesh generator creates a masonry pattern, in the *xy* plane, for eight-noded plane stress elements and six-noded line interface elements. Presently, this is an external, interactive module. The complete source code is given in Appendix B.

A typical example of the screen output of the program with the list of data that the user must provide (in bold) is the following:

screen

```
make_wall diana.dat ↵
Title: Example of pre-processing ↵
 Cracks in the middle of the units? [0/1]
1 ↵
 Interface in the bottom of first course? [0/1]
1 ↵
 Interface in the top of last course? [0/1]
1 ↵
 Each course contains an integer number of units? [0/1]
1 ↵
 First course starts with full unit? [0/1]
1 ↵
 Number of courses?
6 ↵


 Number of complete units per course?
4 ↵
 Number of x divisions per unit?
 (must be even number)
4 ↵
```

```
 Number of y divisions per unit?
2 ↵
 Dimension x of unit?
100. ↵
 Dimension y of unit?
50. ↵
 Fake half_thickness of joints?
0.0 ↵
Fake half_thickness of cracks?
0.0 ↵

*** CREATED FILE diana.dat ***
```

The command `make_wall diana.dat` indicates to create a data file with the name `diana.dat`. The data required by the program are (0 - No, 1 - Yes):

1. `Title:`

   The title of the analysis.

2. `Cracks in the middle of the units? [0/1]`

   Whether potential vertical cracks in the middle of the units are to be included in the model.

3. `Interface in the bottom of first course? [0/1]`

   Whether a masonry joint is to be included in the bottom of the model.

4. `Interface in the top of last course? [0/1]`

   Whether a masonry joint is to be included in the top of the model.

5. `Each course contains an integer number of units? [0/1]`

   Choose between two possible course configurations, see Figure 6.

6. `First course starts with full unit? [0/1]`

   Whether the first (bottom) course starts with a full unit oh half unit, for both configurations of Figure 6.

7. `Number of courses?`

   The number of masonry courses in the model, see Figure 6.

8. `Number of complete units per course?`

   The number of complete units per course, see Figure 6.

9. `Number of x divisions per unit?  (must be even number)`

   The number of division (finite elements) per unit in the *x* direction, see Figure 7.

10. `Number of y divisions per unit?`

The number of division (finite elements) per unit in the y direction, see Figure 7.

11. `Dimension x of unit?`

    The width of the unit added with the thickness of the mortar joint, see Figure 7.

12. `Dimension y of unit?`

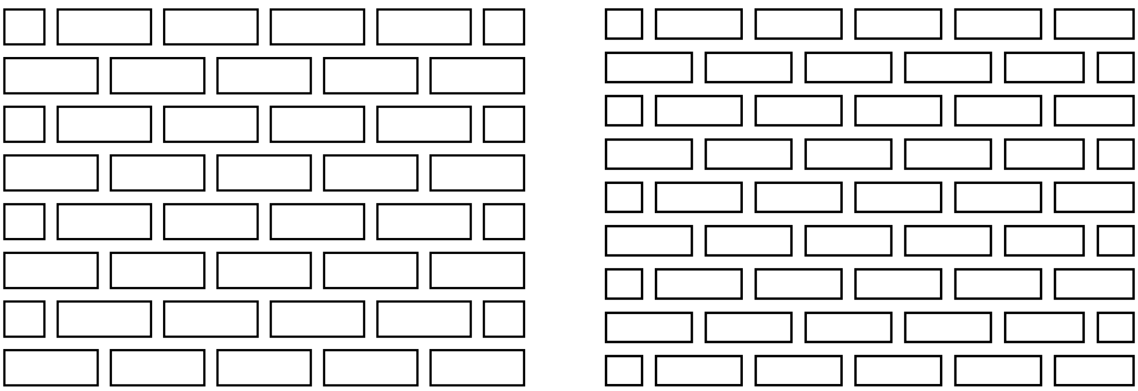
    The height of the unit added with the thickness of the mortar joint, see Figure 7.

13. `Fake half_thickness of joints?`

    A possible (virtual) thickness given to the interfaces to allow easier later manipulation or to obtain nicer graphical output, see Figure 7. The value 0.0 represents the true interface thickness and, even if a different value is provided, a *non-zero thickness is not included in the analysis*. This means that the (fake) dimensions of the unit must always represent the (real) dimensions of the unit plus the (real) thickness of the joints.

14. `Fake half_thickness of cracks?`

    A possible (virtual) thickness given to the interfaces to allow easier later manipulation, see Figure 7. For novice users the value 0.0 is suggested.



(a) (b)

Figure 6 - Examples of input: (a) courses with an integer number of units, wall with eight courses and first course starts with a full unit ; (b) courses without an integer number of units, wall with nine courses and first course does not start with a full unit.

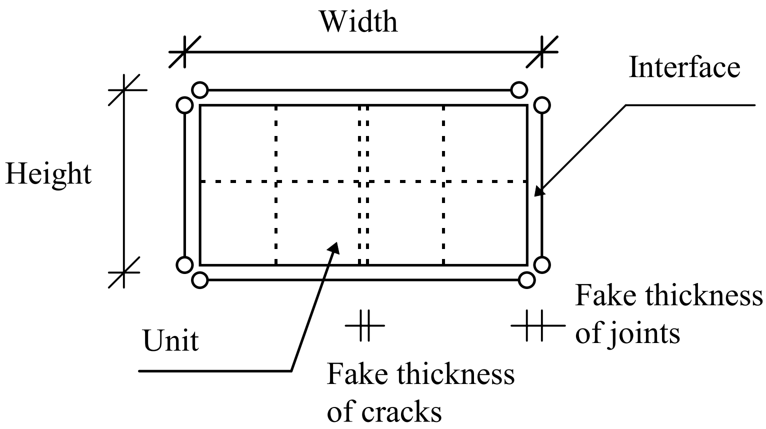


Figure 7 - Detail of a single unit and neighboring interfaces: Four divisions (elements) along the *x* direction and two divisions (elements) along the *y* direction. *The number of elements along the x axis must be even for proper connection with neighboring units*.

The data file automatically generated with the given input is partially shown below. It is noted that four groups have been generated `UNIT, JBED, JHEAD and UCRAC` for the unit elements, bed joint elements, head joint elements and elements representing the potential cracks in the units. A picture of these groups is given in Figure 8. A Gauss 2 × 2 and a Lobatto 3-point integration schemes are suggested for the continuum and interface elements, respectively.

`diana.dat`

```
Example of pre-processing
'COORDINATES'
    1 0.0000000E+00 0.0000000E+00 0.0000000E+00
    2 0.1250000E+02 0.0000000E+00 0.0000000E+00
    3 0.2500000E+02 0.0000000E+00 0.0000000E+00
    4 0.3750000E+02 0.0000000E+00 0.0000000E+00
    5 0.5000000E+02 0.0000000E+00 0.0000000E+00
```

1078 lines skipped

```
 1084 0.3500000E+03 0.3000000E+03 0.0000000E+00
 1085 0.3625000E+03 0.3000000E+03 0.0000000E+00
 1086 0.3750000E+03 0.3000000E+03 0.0000000E+00
 1087 0.3875000E+03 0.3000000E+03 0.0000000E+00
 1088 0.4000000E+03 0.3000000E+03 0.0000000E+00
'GROUPS'
 ELEMEN
  1  UNIT  / 1-192 /
  2  JBED  / 193-304 /
  3  JHEAD / 305-346 /
  4  UCRAC / 347-388 /

'ELEMENTS'
CONNECT
    1 CQ16M    41    42    43    82   107   106   105    81
    2 CQ16M    43    44    45    83   109   108   107    82
```

```
    3 CQ16M    46    47    48    85   112   111   110    84
                                           186 lines skipped
  190 CQ16M   977   978   979  1005  1043  1042  1041  1004
  191 CQ16M   980   981   982  1007  1046  1045  1044  1006
  192 CQ16M   982   983   984  1008  1048  1047  1046  1007
  193 CL12I     1     2     3    41    42    43
  194 CL12I     3     4     5    43    44    45
  195 CL12I     6     7     8    46    47    48
                                           190 lines skipped
  386 CL12I   965   997  1029   964   996  1028
  387 CL12I   911   939   975   910   938   974
  388 CL12I   975  1003  1039   974  1002  1038
MATERI
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 3
  / UCRAC / 4
DATA
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 2
  / UCRAC / 2
GEOMET
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 2
  / UCRAC / 2
'MATERI'
  1   YOUNG  1.D+4
      POISON 0.2
  2   DSTIF  1.D+3 1.D+3
  3   DSTIF  1.D+3 1.D+3
  4   DSTIF  1.D+6 1.D+6
'GEOMET'
  1   THICK  1.D+2
  2   CONFIG MEMBRA
      ZAXIS  0. 0. 1.
      THICK  1.D+2
'DATA'
  1   NGAUS  2 2
  2   NLOBAT 3
'END'
```

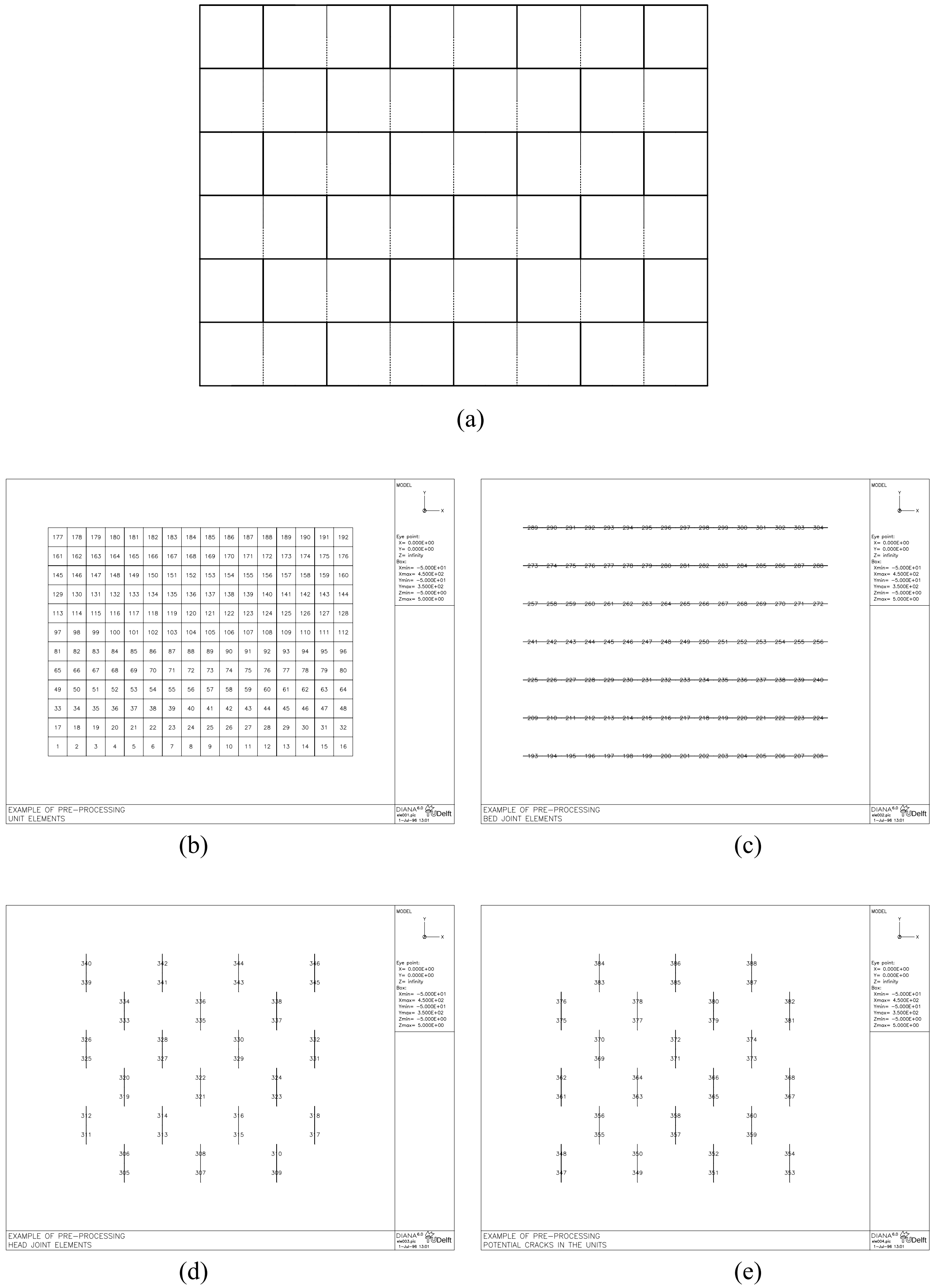


Figure 8 - Automatically generated mesh for (a) masonry wall: (b) unit elements; (c) bed joint elements; (c) head joint elements and (d) potential cracks in the units.

3. Recommendations about input data

The research on masonry has been scarce when compared with other structural materials and experimental data which can be used as input for the model is limited. In the following, some results available in the literature are presented and some recommendations for the use of the model are given.

3.1 Tension mode

The parameters needed are the bond tensile strength f_t and the bond fracture energy G_f^I. The factors that affect the bond between unit and mortar are highly dependent on the units (material, strength, perforation, size, air dried or pre-wetted, etc.), on the mortar (composition, water contents, etc.) and on workmanship (proper filling of the joints, vertical loading, etc.).

For the materials normally used in the Netherlands the following values have been obtained, CUR (1994),

Table 1 - Material data for tension

Unit/mortar combination[a]	f_t (N/mm^2)	G_f^I (Nmm/mm^2)
VE.B	0.33	0.010
VE.C	0.32	0.013
JG.B	0.51	0.018
JG.C	0.87	0.006
KZ.B	0.03	-[b]
KZ.C	0.10	-[b]

[a] VE denotes the Vijf Eiken soft mud clay unit, JG denotes the Joost Gele wire cut clay unit, B denotes 1:2:9 mortar (cement:lime:sand per volume) and C denotes 1 : ½ : 4 ½ mortar.

[b] not available.

It is clear that a recommendation for the value of the bond tensile strength based on the unit type or mortar type is impossible. Nevertheless, an average value of the bond mode I fracture energy equal to 0.012 Nmm/mm^2 can be, in principle, adopted.

3.2 Shear mode

The parameters needed for most applications are the bond strength *c*, the friction angle measured by $\tan\phi$ and the dilatancy angle measured by $\tan\psi$. More advanced applications of the model may require variable friction and dilatancy angles in which case the three additional material parameters indicated in Section 2.1 must be provided. It is noted that a variable friction angle yields a non-constant mode II fracture energy, see Lourenço (1996). In the following, it is assumed that the friction and dilatancy angles can be considered constant.

The bond shear strength depends also on the influence factors given in the Section 3.1. For the materials normally used in the Netherlands the following values have been obtained, CUR (1994),

Table 2 - Material data for shear

Unit/mortar combination	c (N/mm^2)	G_f^{II} (Nmm/mm^2)
VE.B	0.65	0.065 [a]
VE.C	0.85	0.053 [a]
JG.B	0.88	0.129 [a]
JG.C	1.85	0.134 [a]
KZ.B	0.15	0.013 [a]
KZ.C	0.28	0.031 [a]

[a] the bond mode II fracture energy depends on the level of the normal stress which is only partially accounted for in the model.

It can be seen that an average ratio between the bond shear strength and the bond tensile strength cannot be established, see also CUR (1994). Nevertheless, an average value for the bond mode II fracture energy of $\frac{1}{10}c$ can be, in principle, adopted.

Independently from the type of unit or mortar, the value of 0.75 can be adopted for the tangent of the friction angle, CUR (1994) and Atkinson *et al.* (1988). Similarly, a value of 0.0 can be adopted for the tangent of the dilatancy angle, Lourenço (1996).

3.3 Cap mode

The parameters needed for the cap mode are the masonry uniaxial compressive strength f_m, the shape of the elliptical cap given by C_s, the compressive fracture energy G_{fc} and the equivalent relative displacement κ_p which corresponds to the uniaxial compressive strength.

It is a current practice to define the uniaxial compressive strength of masonry from the uniaxial compressive strengths of the unit and mortar, see e.g. EuroCode 6 (CEN, 1995). For the materials normally used in the Netherlands the following values have been obtained, CUR (1994),

Table 3 - Material data for compression

Unit/mortar combination	f_m (N/mm^2)
VE.C	8.7 [a]
JG.C	16.3 [a]
KZ.C	20.0 [a]

[a] these values were obtained with a stack bond prism. Different compressive strengths should be adopted for the head and bed joints to simulate anisotropic behavior.

For the parameter C_s, a value of 9.0 can be adopted, Lourenço (1996).

Not much data exists about the post-peak behavior of masonry prisms. For an estimate of the compressive fracture energy, it is recommended to adopted the values proposed for concrete in the Model Code 90 (CEB-FIP, 1991), see Figure 9. The equivalent relative displacement κ_p, can be calculated so that the total masonry strain equals 0.2 %, see EuroCode 6.

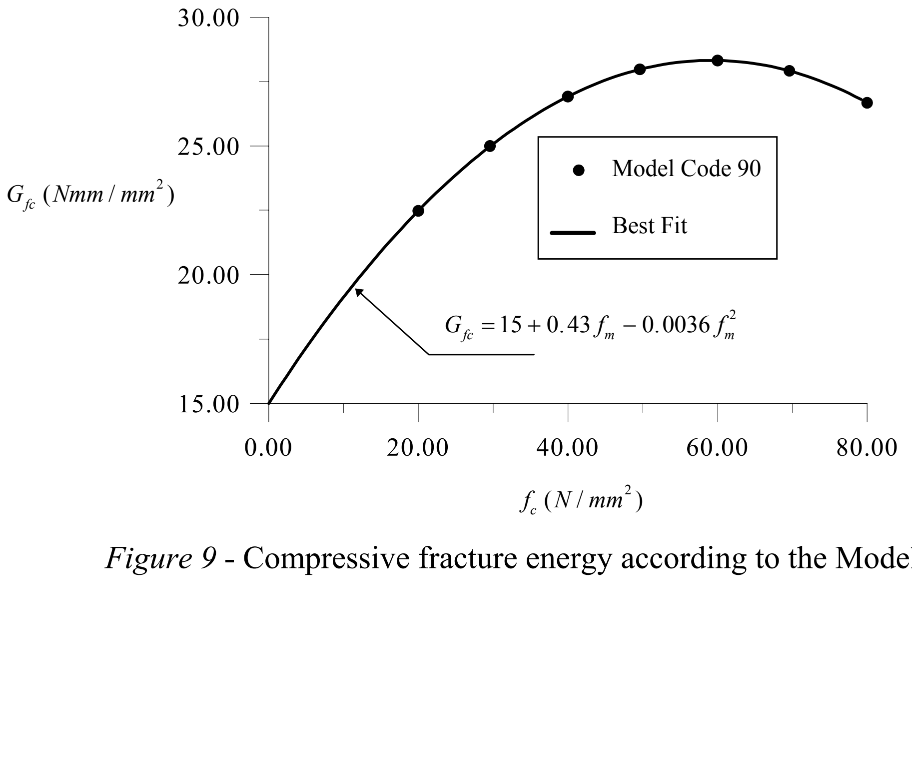


Figure 9 - Compressive fracture energy according to the Model Code 90

3.4 Example

The values suggested to use as input of the model, in the case of masonry made with units and normal strength mortar currently used in the developed countries, are condensed below.

file.dat

```
'MATERI'
     1  DSTIF   kn kt
        MASINT
        GAPVAL  ft
        MODE1   4
        MO1VAL  0.012
        FRCVAL  c 0.75 0.0
        MODE2   4
        MO2VAL  c/10
        CAPVAL  fm 9.0
        MODEC   5
        MOCVAL  Gfc kp
```

The interface elastic stiffness values can be calculated from the thickness of the joint h_j, the Young's moduli of unit and joint, E_u and E_j, respectively, and the shear moduli of unit and joint, G_u and G_j, respectively, as, CUR(1994),

$$k_n = \frac{E_u E_j}{h_j \left(E_u - E_j \right)}$$

$$k_t = \frac{G_u G_j}{h_j \left(G_u - G_j \right)}$$

The different strength values f_t, c and f_m are given from the available experimental data or tests in collected samples. The compressive fracture energy is given by, see Figure 5,

$$G_{fc} = 15 + 0.43 f_m - 0.0036 f_m^2$$

and the equivalent relative displacement reads

$$\kappa_p = \left\{ 0.002 - f_m \left[\frac{1}{E_u} + \frac{1}{k_n (h_u + h_j)} \right] \right\} f_m$$

in order to obtain a masonry strain of 0.2 % at peak stress.

The input parameters for the model are, therefore:

- the elastic properties of the interface

  k_n and k_t (**or** E_u, G_u, ν_u and E_j, G_j, ν_j and h_j)

- tensile strength of the joint

  f_t

- shear strength of the joint (cohesion)

  c

- compressive strength of masonry

  f_m

4. Example 1 - Masonry pier with point load

This example illustrates the use of DIANA for the analysis of a masonry pier with a point load, see Figure 10. In the center of the pier a splitting crack arises which propagates in a catastrophic manner after peak load. The computed crack path is straight and vertical indicating that the crack jumps from head joint to head joint right through the unit. The chosen material parameters represent the inelastic behavior according to Chapter 3.

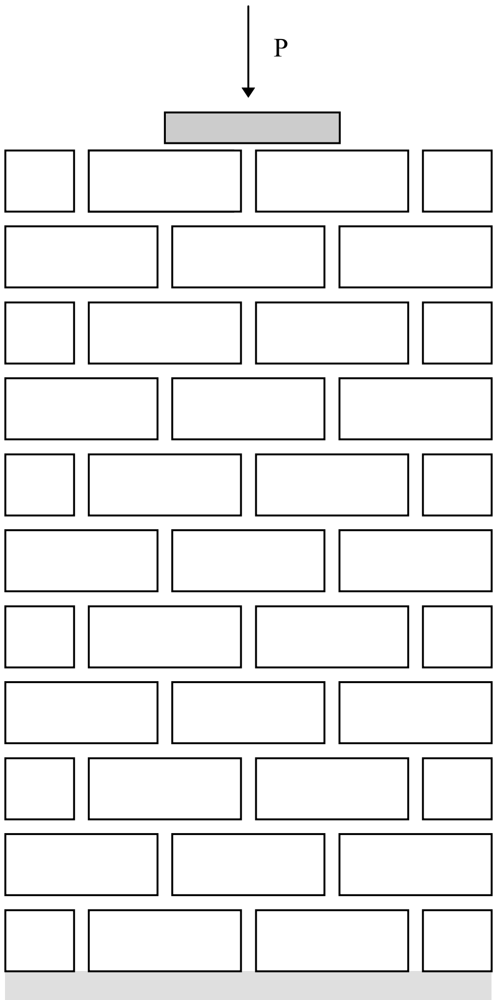


Figure 10 - Masonry pier with point load

4.1 Mesh generation

The full mesh is generated using the external masonry pre-processor `make_wall`. The run-command is `make_wall mesh.dat` and the interactive input data are shown below. The units are 140 × 50 × 100 mm^3 and the joints are 10 mm thick. Note that a non-zero fake_interface thickness is used to avoid showing interpenetration of the units. This is a representation of the true thickness of the joint that is not taken into account in the analysis. In reality, the interfaces have zero thickness and the dimensions of the continuum elements must be enlarged by the unit joint thickness.

Input data *screen*

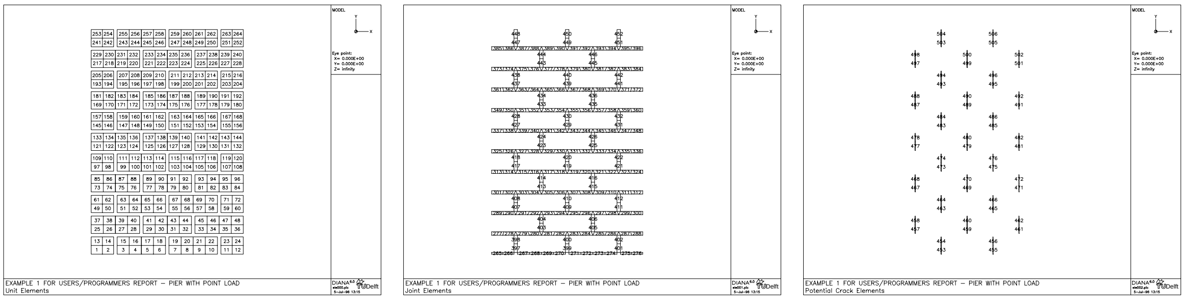
```
make_wall mesh.dat ↵
Title:Example1 for users/programmers report - Pier with point load ↵
 Cracks in the middle of the units? [0/1]
1 ↵
 Interface in the bottom of first course? [0/1]
1 ↵
 Interface in the top of last course? [0/1]
0 ↵
 Each course contains an integer number of units? [0/1]
1 ↵
 First course starts with full unit? [0/1]
0 ↵
 Number of courses?
11 ↵
 Number of complete units per course?
3 ↵
 Number of x divisions per unit?
 (must be even number)
4 ↵
 Number of y divisions per unit?
2 ↵
 Dimension x of unit?
150. ↵
 Dimension y of unit?
60. ↵
 Fake half_thickness of joints?
5.0 ↵
 Fake half_thickness of cracks?
0.0 ↵
*** CREATED FILE mesh.dat ***
```

The external mesh generator produces a file `mesh.dat`. The generated mesh is evaluated with the graphic output device of Module POST. The command file shown below produces plot files ele000.pic, ele001.pic and ele002.pic, with the mesh and element numbers for the units, joints and potential cracks in the units, respectively. The run command is

diana mesh.dat mesh.com

Commands `mesh.com`

```
*FILOS
 INITIA
*INPUT
*POST
 MODEL
    SELECT ELEMEN UNIT /
 END MODEL
 LAYOUT
   ELEMEN.N
 END LAYOUT
 OUTPUT GRAPHI FI="ele"
   TEXT "Unit Elements"
 END OUTPUT
 MODEL
    SELECT ELEMEN JBED JHEAD /
 END MODEL
 OUTPUT GRAPHI FI="ele"
    TEXT "Joint Elements"
 END OUTPUT
 MODEL
    SELECT ELEMEN UCRAC /
 END MODEL
 OUTPUT GRAPHI FI="ele"
    TEXT "Potential Crack Elements"
 END OUTPUT
*END
```



(a) (b) (c)

Figure 11 - Generated mesh:

(a) Unit elements; (b) joint elements; (c) potential unit crack elements.

The file `mesh.dat` contains the node coordinates and element connectivity. This file is to be completed with the material and loading data. The material parameters correspond to the masonry behavior described in Chapter 3. At the top of the pier, the force is applied with an infinitely rigid plate, modeled by tying the vertical displacement of the nodes under the plate. At the bottom of the pier, the (mid-) nodes belonging to the symmetry axis are also tied to have symmetric horizontal displacements. The completed data file for the analysis `example1.dat` is shown below.

`example1.dat`

```
Example 1 for users/programmers report - Pier with point load
'COORDINATES'
    1 0.5000000E+01 0.0000000E+00 0.0000000E+00
    2 0.2125000E+02 0.0000000E+00 0.0000000E+00
    3 0.3750000E+02 0.0000000E+00 0.0000000E+00
    4 0.5375000E+02 0.0000000E+00 0.0000000E+00
                                                                1408 lines skipped
 1413 0.3962500E+03 0.6550000E+03 0.0000000E+00
 1414 0.4125000E+03 0.6550000E+03 0.0000000E+00
 1415 0.4287500E+03 0.6550000E+03 0.0000000E+00
 1416 0.4450000E+03 0.6550000E+03 0.0000000E+00
'GROUPS'
 ELEMEN
  1  UNIT  / 1-264 /
  2  JBED  / 265-396 /
  3  JHEAD / 397-452 /
  4  UCRAC / 453-506 /
'ELEMENTS'
CONNECT
    1 CQ16M    31    32    33    62    81    80    79    61
    2 CQ16M    33    34    35    63    83    82    81    62
    3 CQ16M    36    37    38    65    86    85    84    64
    4 CQ16M    38    39    40    66    88    87    86    65
                                                                156 lines skipped
  261 CQ16M  1359  1360  1361  1382  1409  1408  1407  1381
  262 CQ16M  1361  1362  1363  1383  1411  1410  1409  1382
  263 CQ16M  1364  1365  1366  1385  1414  1413  1412  1384
  264 CQ16M  1366  1367  1368  1386  1416  1415  1414  1385
  265 CL12I     1     2     3    31    32    33
  266 CL12I     3     4     5    33    34    35
  267 CL12I     6     7     8    36    37    38
  268 CL12I     8     9    10    38    39    40
                                                                130 lines skipped
  503 CL12I  1301  1327  1349  1300  1326  1348
  504 CL12I  1349  1375  1397  1348  1374  1396
  505 CL12I  1311  1333  1359  1310  1332  1358
  506 CL12I  1359  1381  1407  1358  1380  1406
MATERI
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 2
  / UCRAC / 3
DATA
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 2
  / UCRAC / 2
GEOMET
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 2
  / UCRAC / 2
:                    Up to here this file was produced by the
:                    diana’s external masonry mesh generator
'MATERI'
  1   YOUNG  15000.
      POISON 0.2
:           hj = 10 mm
:           Eu = 15000.     Gu = 15000 / 2 / ( 1 + 0.2 ) = 6250
:           Ej =  6000.     Gj =  6000 / 2 / ( 1 + 0.2 ) = 2500
:           kn = 15000 * 3000 / ( 10 * ( 15000 - 3000 ) ) = 1000
:           ks =  6250 * 2500 / ( 10 * (  6250 - 2500 ) ) = 417
:           ft = 0.15            c = 0.3            fm = 12.
```

```
:           Gfc = 15 + 0.43 * 12 - 0.0036 * 12 * 12 = 20
:           kp  = 12*(0.002-12*(1/15000+1/1000/(10.+50.))=0.001
  2   DSTIF  1000. 417.
      MASINT
      GAPVAL 0.15
      MODE1  4
      MO1VAL 0.012
      FRCVAL 0.3 0.75 0.0
      MODE2  4
      MO2VAL 0.03
      CAPVAL 12. 9.0
      MODEC  5
      MOCVAL 20. 0.001
  3   DSTIF  1.D+6 1.D+6
      DISCRA 1
      DCRVAL 0.75
      MODE1  2
      MO1VAL 0.025
'GEOMET'
  1   THICK  1.D+2
  2   CONFIG MEMBRA
      ZAXIS  0. 0. 1.
      THICK  1.D+2
'DATA'
  1   NGAUS  2 2
  2   NLOBAT 3
'TYINGS'
EQUAL TR 2
: Simulates infinitely rigid plate
/ 1397-1405 / 1406
: Pair of nodes at the bottom of the pier
: with symmetric horizontal displacement
FIX TR 1
 15  16 TR 1 -1.0
'SUPPOR'
: Bottom of pier
/ 1-30 / TR 2
'LOADS'
CASE 1
NODAL
: Vertical force of 1000 N
 / 1397-1406 / F 2 -125.0
'END'
```

An additional DIANA-job, with the file `supload.com`, generates the graphic output file `supload000.pic` to check the loads and supports. The run command is

`diana example1.dat supload.com`

Commands `supload.com`

```
*FILOS
 INITIA
*INPUT
*ELASSE
*POST
 MODEL
   SELECT ELEMEN UNIT JBED /
   HIDDEN
```

```
 END MODEL
 LAYOUT
   MODEL.E
 END LAYOUT
 OUTPUT GRAPHI FI="loa"
  LOAD
  SUPPOR
  TEXT "Check Loads and Supports. Plot Edges only"
 END OUTPUT
*END
```

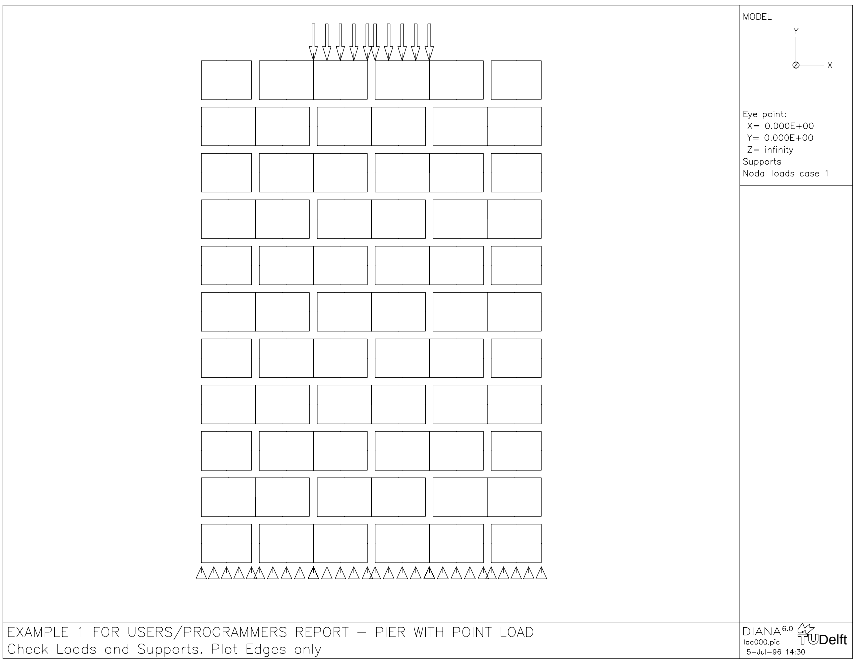


Figure 12 - Supports and loads for masonry pier with point load

4.2 Nonlinear analysis

It is necessary that the nonlinear analysis starts after a linear static analysis where most checks about the model are performed and the global stiffness matrix is assembled for the first time. Also a series of commands are necessary to define the types of nonlinearity in the model and the combinations of loads applied to the model. Then, we run this initial job

`diana example1.dat non.com`

Commands non.com

```
*FILOS
 INITIA
*INPUT
*ELASSE
*ELMAT
*LOADS
```

```
*ORDER
*SOLVE
*NONLIN
 SEGMENT INITIA/IN30LL
 INITIA
  ANALYS PHYSIC
  OPTION TANGEN NONSYM
  USE
     INTERF
  END USE
 END INITIA
 LOADIN
  LOAD(1): (1) 1.0 /
 END LOADIN
*END
```

The nonlinear analysis is performed under the special interface arc-length procedure, `SELECT ARCLEN INTTOT`. We execute a first load step of 20.0 (= 20 kN), which is still completely linear. This is followed by eight load steps of 10.0, four steps of 2.5 and five steps of 0.5 up to peak load. The (small) critical steps close to a sharp snap-back are controlled with the maximum incremental / total relative displacement (`SELECT ARCLEN INTREL`) in the interface. Afterwards, we return to the standard `INTTOT` arc-length control. The unloading is traced with ten steps of 5.0 and twenty steps of 1.0. After each run, we run Module POST, to make plots of the deformed structure, the tensile principal stresses and the normal opening of the joints as a representation of cracking in the joints and units.

The first step, is performed with the run command

`diana example1.dat nonlin1.com`

Commands nonlin1.com

```
*NONLIN/CI33LL
 SEGMENT EXECUT/XQ31LL
 SELECT
  NODES 1405 /
  ELEMENT NONE //
  END ELEMEN
 END SELECT
 OUTPUT TABULA NONLIN
  DISPLA TOTAL GLOBAL
 END OUTPUT
 SELECT ARCLEN INTTOT
 END SELECT ARCLEN
 EXECUTE LOAD(1) STEPS
  LINE SEARCH
  SIZE.P 20.0(1) /
  PERFOR.R NEWTON REGULA MI=15
  NORM ENERGY NEWREF CONTIN CO=1.d-6 AB=1.d+4
 END EXECUTE STEPS
*POST/CI30LL
```

```
 SEGMENT WRITE/WR30LL
 MODEL
  SELECT ELEMEN UNIT /
  HIDDEN
 END MODEL
 LAYOUT
  MODEL.E LI=..
  ELEMEN.
  NODES. DATA.E LI=-
 END LAYOUT
 OUTPUT GRAPHI NONLIN FI="i"
  TEXT "Incremental displacements"
  DISPLA INCREM
 END OUTPUT
MODEL
  SELECT ELEMEN UNIT /
  HIDDEN
 END MODEL
 LAYOUT
  MODEL.E LI=-
  ELEMEN. DATA.F
 END LAYOUT
 OUTPUT GRAPHI NONLIN FI="s"
  TEXT "Tensile principal stresses"
  STRESS TOTAL PRINCI XX YY INTPNT FR=0.D0
 END OUTPUT
 LAYOUT
  MODEL.E LI=-
  ELEMEN. DATA.F6
 END LAYOUT
 OUTPUT GRAPHI NONLIN FI="u"
  TEXT "Relative opening displacement normal to interface"
    STRAIN FORCE N INTPNT FR=0.d0
 END OUTPUT
*END
```

We can continue the nonlinear analysis using almost identical command files. For the next two command files, we just change the `SIZE` line in the `EXECUTE LOAD(1) STEPS` block as given below. The run commands are

`diana example1.dat nonlin2.dat`

`diana example1.dat nonlin3.dat`

Commands nonlin2.com

```
*NONLIN/CI33LL
```

14 lines skipped

```
SIZE.P 10.0(8) /
```

3 lines skipped

```
*POST
:    like nonlin1.com
```

16 lines skipped

Commands nonlin3.com

```
*NONLIN/CI33LL
                                                    14 lines skipped
SIZE.P 2.5(4) /
                                                     3 lines skipped
*POST
:    like nonlin1.com
                                                    16 lines skipped
```

A very sharp snap-back is obtained at peak load due to the sudden energy release in the straight crack that arises under the load. For the next steps, we must also change the `SELECT ARCLEN` line as given below. The run command is

`diana example1.dat nonlin4.dat`

Commands `nonlin4.com`

```
*NONLIN/CI33LL
                                                    11 lines skipped
 SELECT ARCLEN INTREL
 END SELECT ARCLEN
 EXECUTE LOAD(1) STEPS
    LINE SEARCH
    SIZE.P 0.5(5) /
    PERFOR.R NEWTON REGULA MI=15
    NORM ENERGY NEWREF CONTIN CO=1.d-6 AB=1.d+4
 END EXECUTE STEPS
*POST
:    like nonlin1.com
                                                    16 lines skipped
```

After peak load, it is possible to return to the more stable arc-length procedure where the maximum incremental relative displacement is adopted as control parameter. For the next steps, we change again the `SELECT ARCLEN` line as given below. The run command is

`diana example1.dat nonlin5.dat`

Commands `nonlin5.com`

```
*NONLIN/CI33LL
                                                    11 lines skipped
 SELECT ARCLEN INTTOT
 END SELECT ARCLEN
 EXECUTE LOAD(1) STEPS
    LINE SEARCH
    SIZE.P 5.0(10) 1.0(20) /
    PERFOR.R NEWTON REGULA MI=15
    NORM ENERGY NEWREF CONTIN CO=1.d-6 AB=1.d+4
 END EXECUTE STEPS
*POST
:    like nonlin1.com
                                                    16 lines skipped
```

After each load step, the selected data for node `1405` is written to the output file `diana.tb` as specified in the `SELECT` block. A selection of the tabulated output of the last load step is given below.

Nonlinear output nonlin5.com

```
 Analysis type       : NONLIN
 Step nr.            :    48
 Load factor         :   0.4564E+02
 Result              : DISPLA TOTAL  TRANSL
 Axes                : GLOBAL


    Nodes          TDtX        TDtY        TDtZ
     1405   -0.8003E-02 -0.1123E+00  0.0000E+00
```

4.3 Interpretation of the results

From the tabulated output, it is possible to obtain a force-displacement diagram with the `scan` filter. The run command is

`scan < diana.tb >> xy.dat`

Filter scan

```
awk 'BEGIN { displa_line=10;
             f = "%15.4e %15.4e\n"
             printf f, 0.0, 0.0
           }
/TDtX/    { displa_line = 0 }
{
displa_line += 1
if ( displa_line == 2 ) { disp = $3 }
if ( displa_line == 8 ) { force = $4
printf f, -disp, force }
}
```

The force-displacement diagram for the rigid plate is given in Figure 13. The rising portion of the curve appears to be almost linear, indicating that the effect of cracking prior to reaching the maximum load is negligible. After reaching the maximum load, a very sudden decrease of both the load and the displacement occurs. The other figures show the plots produced after steps 1, 9, 13, 18 and 48. For each load step, we plot the incremental deformed mesh, plotted with a solid line, the tensile principal stresses at the integration points, plotted with a solid line, and the normal opening of all interface elements, plotted with a variable thickness solid line.

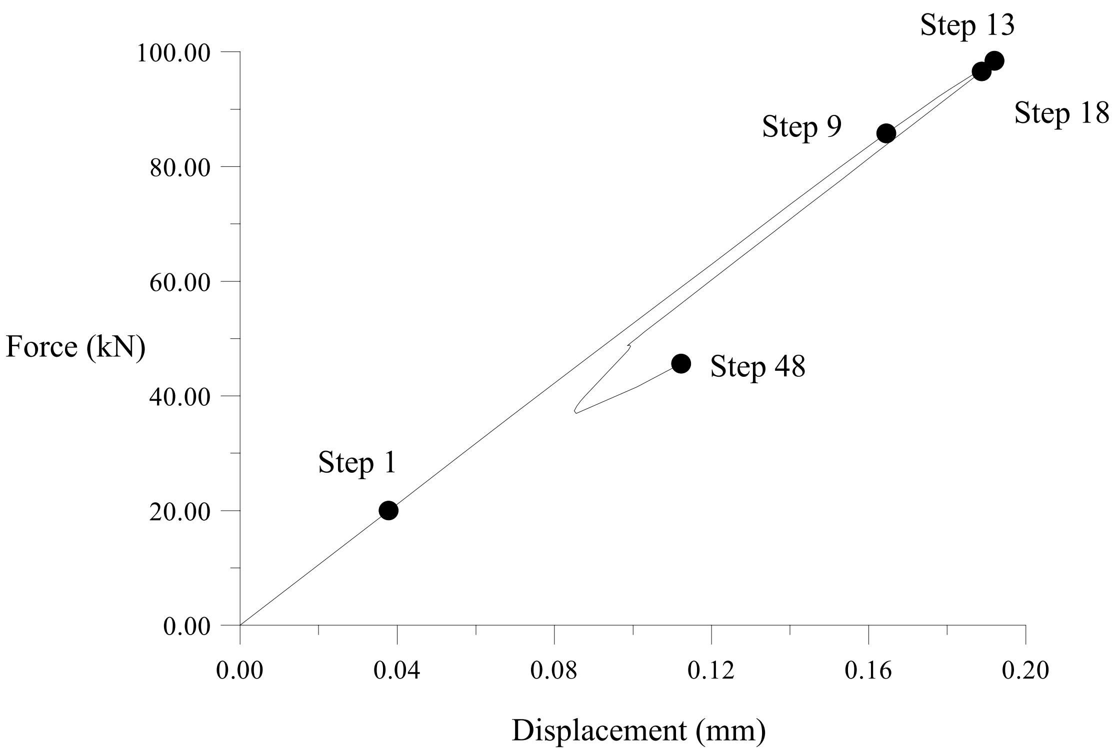
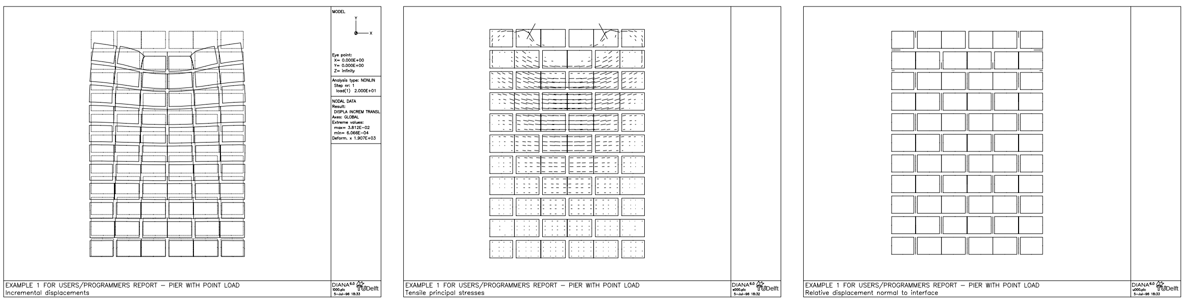
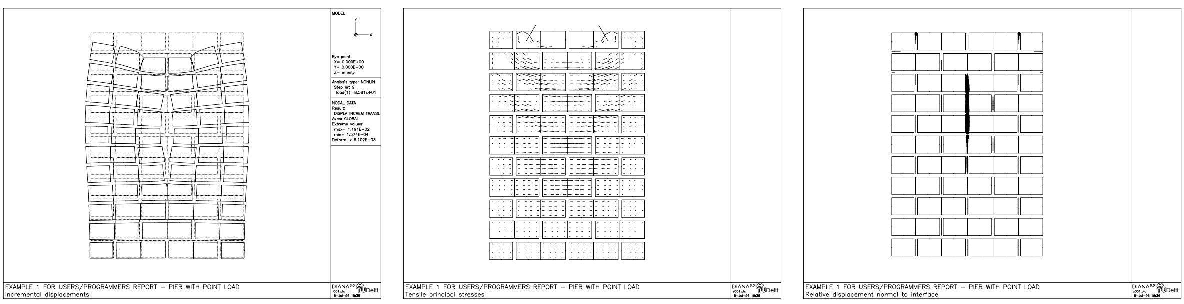


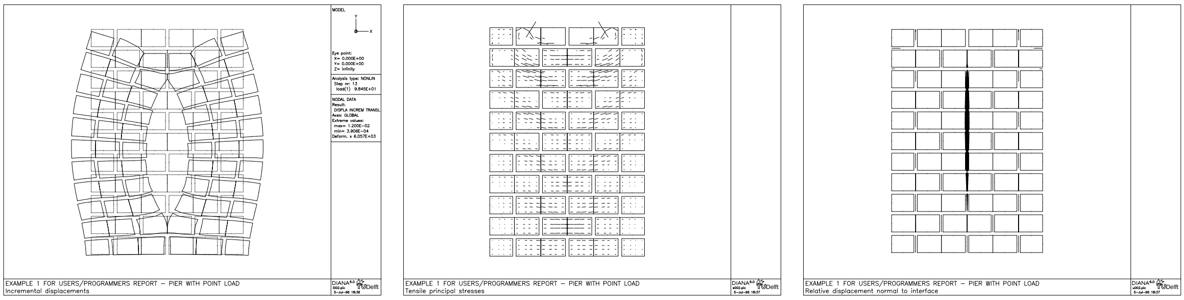
Figure 14 - Force-displacement diagram for masonry pier with point load



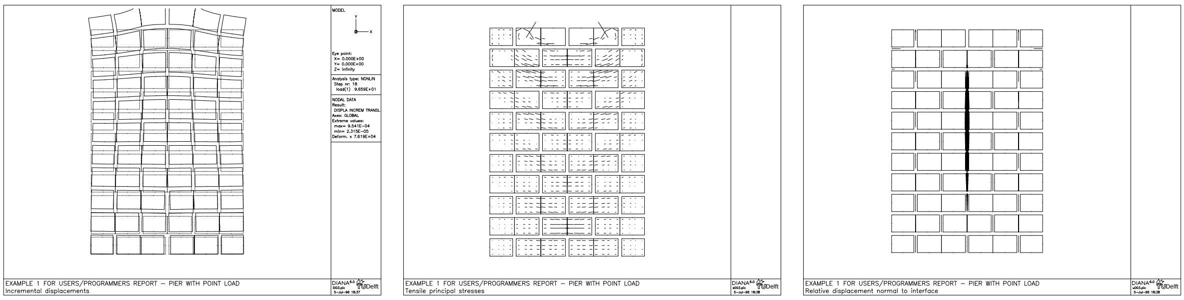
Step 1



Step 9

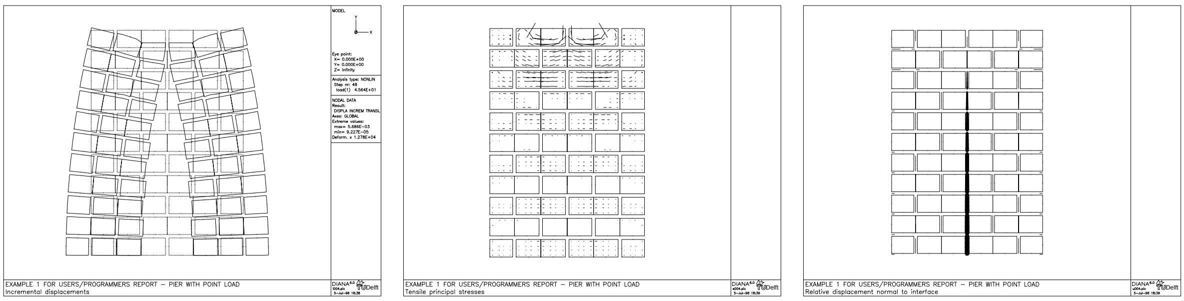


Step 13



Step 18

Figure 15 - Plots for masonry pier with point load: incremental deformed mesh, tensile principal stresses and normal opening of interfaces (cont.).



Step 48

Figure 15 - Plots for masonry pier with point load: incremental deformed mesh, tensile principal stresses and relative normal opening of interfaces (contd.).

5. Example 2 - Masonry shear wall with an opening

This example illustrates the use of DIANA for the analysis of a masonry shear wall with an opening, see Figure 16. An initial vertical load *p* is applied before shearing the wall with the horizontal force *F*. The opening in the center of the wall forces the compressive strut, which arises during loading, to spread around it. This leads to diagonal stepped cracks starting from two corners of the opening and bending cracks at the top and bottom of the smaller piers defined by the opening. At collapse, four rigid blocks are formed.

Potential cracks in the units have not been modeled to increase the legibility of the plots. Nevertheless, potential cracks in the units should always be included in the analysis, Lourenço (1996). The chosen material parameters represent the inelastic behavior according to Chapter 3.

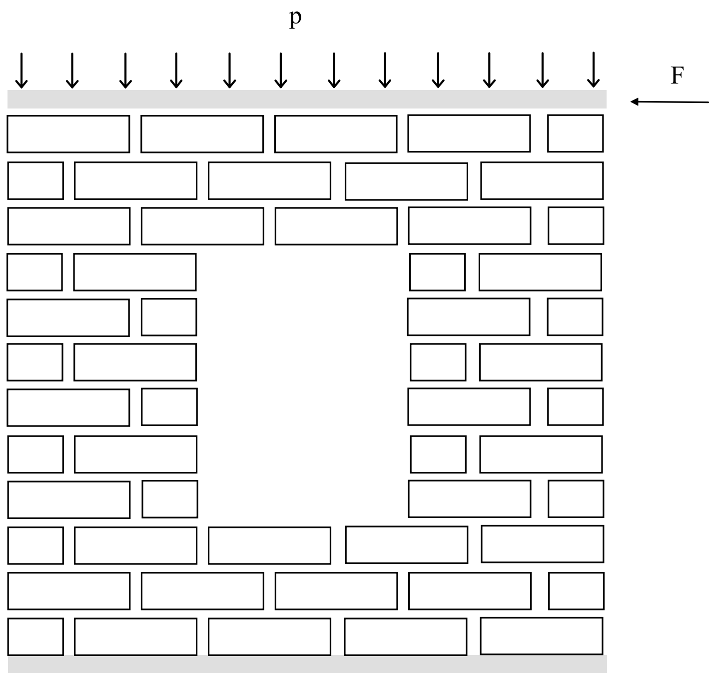


Figure 16 - Masonry pier with point load

5.1 Mesh generation

The full mesh is generated using the external masonry pre-processor `make_wall`. The run-command is `make_wall mesh.dat` and the interactive input data are shown below. The units are 140 × 50 × 100 mm^3 and the joints are 10 *mm* thick. Note that a non-zero fake_interface thickness is used to avoid showing interpenetration of the units.

This is a representation of the true thickness of the joint that is not taken into account in the analysis. In reality, the interfaces have zero thickness and the dimensions of the continuum elements must be enlarged by the unit joint thickness.

Input data *screen*

```
make_wall mesh.dat ↵
Title: Example 2 for users/programmers report - Shear wall with an
opening ↵
 Cracks in the middle of the units? [0/1]
0 ↵
 Interface in the bottom of first course? [0/1]
1 ↵
 Interface in the top of last course? [0/1]
1 ↵
 Each course contains an integer number of units? [0/1]
0 ↵
 First course starts with full unit? [0/1]
0 ↵
 Number of courses?
12 ↵
 Number of complete units per course?
4 ↵
 Number of x divisions per unit?
 (must be even number)
4 ↵
 Number of y divisions per unit?
2 ↵
 Dimension x of unit?
150. ↵
 Dimension y of unit?
60. ↵
 Fake half_thickness of joints?
5.0 ↵
*** CREATED FILE mesh.dat ***
```

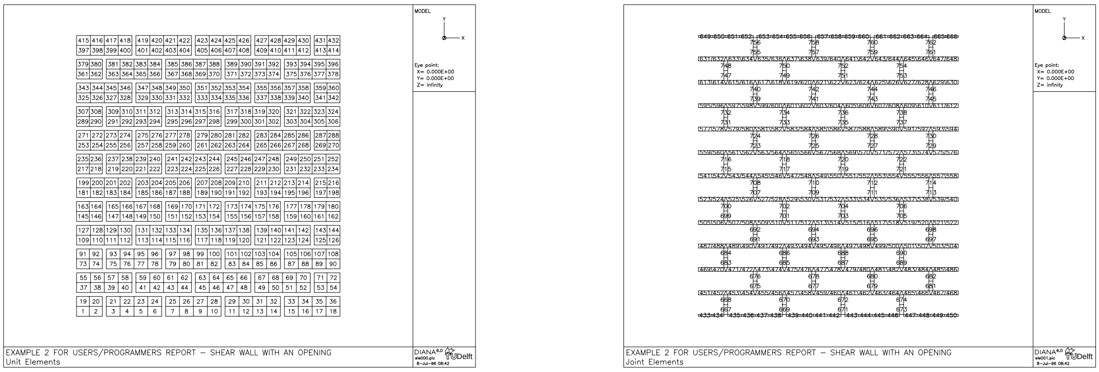
The external mesh generator produces a file `mesh.dat.` The generated mesh is evaluated with the graphic output device of Module POST. The command file shown below produces plot files ele000.pic and ele001.pic, with the mesh and element numbers for the units and joints, respectively. The run command is

`diana mesh.dat mesh.com`

Commands `mesh.com`

```
*FILOS
 INITIA
*INPUT
*POST
 MODEL
```

```
   SELECT ELEMEN UNIT /
 END MODEL
 LAYOUT
   ELEMEN.N
 END LAYOUT
 OUTPUT GRAPHI FI="ele"
   TEXT "Unit Elements"
 END OUTPUT
 MODEL
   SELECT ELEMEN JBED JHEAD /
 END MODEL
 OUTPUT GRAPHI FI="ele"
   TEXT "Joint Elements"
 END OUTPUT
*END
```



(a) (b)

Figure 17 - Generated mesh: (a) Unit elements; (b) joint elements

The file `mesh.dat` contains the node coordinates and element connectivity of a regular masonry pattern. It suffices to comment (**:**) the lines with the elements that have no physical correspondence to form the opening. The file is also to be completed with the material and loading data. The material parameters correspond to the masonry behavior described in Chapter 3. The top and bottom of the wall are assume to remain straight. A vertical force p equal to 1.0 N/mm^2 is applied before shearing the wall with the force F. The completed data file for the analysis `example2.dat` is shown below.

`example2.dat`

```
Example 2 for users/programmers report - Shear wall with an
opening
'COORDINATES'
    1 0.5000000E+01 0.0000000E+00 0.0000000E+00
    2 0.2125000E+02 0.0000000E+00 0.0000000E+00
    3 0.3750000E+02 0.0000000E+00 0.0000000E+00
    4 0.5375000E+02 0.0000000E+00 0.0000000E+00
```

2102 lines skipped

```
 2107 0.6212500E+03 0.7200000E+03 0.0000000E+00
 2108 0.6375000E+03 0.7200000E+03 0.0000000E+00
```

```
 2109 0.6537500E+03 0.7200000E+03 0.0000000E+00
 2110 0.6700000E+03 0.7200000E+03 0.0000000E+00
'GROUPS'
 ELEMEN
  1  UNIT  / 1-432 /
  2  JBED  / 433-666 /
  3  JHEAD / 667-762 /
'ELEMENTS'
CONNECT
    1 CQ16M    42    43    44    84   108   107   106    83
    2 CQ16M    44    45    46    85   110   109   108    84
    3 CQ16M    47    48    49    87   113   112   111    86
    4 CQ16M    49    50    51    88   115   114   113    87
```
126 lines skipped
```
  131 CQ16M   622   623   624   660   688   687   686   659
  132 CQ16M   624   625   626   661   690   689   688   660
:  133 CQ16M   626   627   628   662   692   691   690   661
:  134 CQ16M   628   629   630   663   694   693   692   662
:  135 CQ16M   631   632   633   665   697   696   695   664
:  136 CQ16M   633   634   635   666   699   698   697   665
:  137 CQ16M   635   636   637   667   701   700   699   666
:  138 CQ16M   637   638   639   668   703   702   701   667
  139 CQ16M   640   641   642   670   706   705   704   669
  140 CQ16M   642   643   644   671   708   707   706   670
```
7 lines skipped
```
  148 CQ16M   725   726   727   764   791   790   789   763
  149 CQ16M   727   728   729   765   793   792   791   764
  150 CQ16M   729   730   731   766   795   794   793   765
:  151 CQ16M   732   733   734   768   798   797   796   767
:  152 CQ16M   734   735   736   769   800   799   798   768
:  153 CQ16M   736   737   738   770   802   801   800   769
:  154 CQ16M   738   739   740   771   804   803   802   770
:  155 CQ16M   741   742   743   773   807   806   805   772
:  156 CQ16M   743   744   745   774   809   808   807   773
  157 CQ16M   745   746   747   775   811   810   809   774
  158 CQ16M   747   748   749   776   813   812   811   775
```
272 lines skipped
```
  431 CQ16M  2001  2002  2003  2027  2067  2066  2065  2026
  432 CQ16M  2003  2004  2005  2028  2069  2068  2067  2027
  433 CL12I     1     2     3    42    43    44
  434 CL12I     3     4     5    44    45    46
```
56 lines skipped
```
  491 CL12I   517   518   519   558   559   560
  492 CL12I   519   520   521   560   561   562
:  493 CL12I   522   523   524   562   563   564
:  494 CL12I   524   525   526   564   565   566
:  495 CL12I   526   527   528   567   568   569
:  496 CL12I   528   529   530   569   570   571
:  497 CL12I   531   532   533   571   572   573
:  498 CL12I   533   534   535   573   574   575
  499 CL12I   535   536   537   576   577   578
  500 CL12I   537   538   539   578   579   580
```
258 lines skipped
```
  759 CL12I  1928  1957  1992  1927  1956  1991
  760 CL12I  1992  2021  2056  1991  2020  2055
  761 CL12I  1937  1962  2001  1936  1961  2000
  762 CL12I  2001  2026  2065  2000  2025  2064
:                    Up to here this file was produced by the
:                    diana's external masonry mesh generator
MATERI
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 2
```

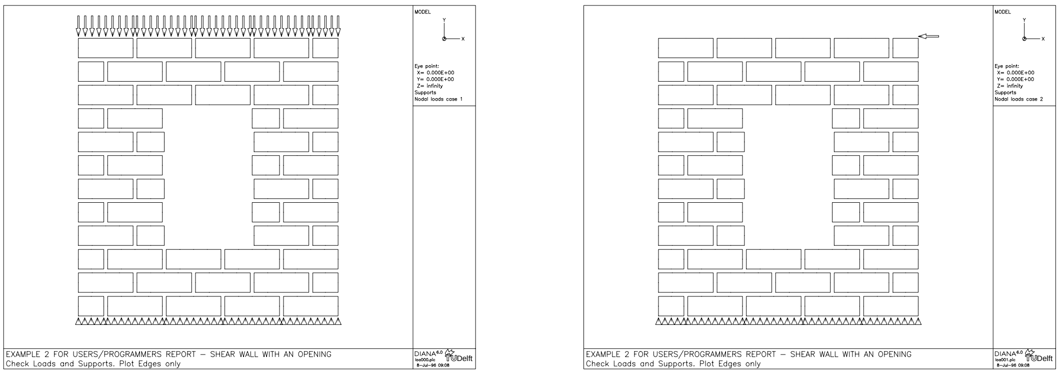
```
DATA
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 2
GEOMET
  / UNIT  / 1
  / JBED  / 2
  / JHEAD / 2
'MATERI'
  1   YOUNG  15000.
      POISON 0.2
:         hj = 10 mm
:         Eu = 15000.       Gu = 15000 / 2 / ( 1 + 0.2 ) = 6250
:         Ej =  6000.       Gj = 6000 / 2 / ( 1 + 0.2 ) = 2500
:         kn = 15000 * 3000 / ( 10 * ( 15000 - 3000 ) ) = 1000
:         ks =  6250 * 2500 / ( 10 * (  6250 - 2500 ) ) = 417
:         ft = 0.15            c = 0.3             fm = 12.
:         Gfc = 15 + 0.43 * 12 - 0.0036 * 12 * 12 = 20
:         kp  = 12*(0.002-12*(1/15000+1/1000/(10.+50.))) = 0.012
  2   DSTIF  1000. 417.
      MASINT
      GAPVAL 0.15
      MODE1  4
      MO1VAL 0.012
      FRCVAL 0.3 0.75 0.0
      MODE2  4
      MO2VAL 0.03
      CAPVAL 12. 9.0
      MODEC  5
      MOCVAL 20. 0.012
'GEOMET'
  1   THICK  1.D+2
  2   CONFIG MEMBRA
      ZAXIS  0. 0. 1.
      THICK  1.D+2
'DATA'
  1   NGAUS  2 2
  2   NLOBAT 3
'TYINGS'
EQUAL TR 1
: Simulates stiff concrete beam
/ 2070-2109 / 2110
EQUAL TR 2
/ 2070-2109 / 2110
'SUPPOR'
: Bottom of wall
/ 1-41 / TR 1 TR 2
'LOADS'
CASE 1
NODAL
: Vertical pressure of 1.0 N/mm2
 / 2070-2110 / F 2 -1646.5
CASE 2
NODAL
: Horizontal force of 1000 N
 / 2110 / F 1 -1000.
'END'
'END'
```

An additional DIANA-job, with the file `supload.com`, generates the graphic output file `supload000.pic` to check the removed elements, loads and supports. The run command is

`diana example2.dat supload.com`

Commands `supload.com`

```
*FILOS
 INITIA
*INPUT
*ELASSE
*POST
 MODEL
   SELECT ELEMEN UNIT JBED /
   HIDDEN
 END MODEL
 LAYOUT
   MODEL.E
 END LAYOUT
 OUTPUT GRAPHI FI="loa"
  LOAD
  SUPPOR
  TEXT "Check Loads and Supports. Plot Edges only"
 END OUTPUT
*END
```



(a) (b)

Figure 18 - Supports and loads for masonry shear wall: (a) load case 1; (b) load case 2

5.2 Nonlinear analysis

It is necessary that the nonlinear analysis starts after a linear static analysis where most checks about the model are performed and the global stiffness matrix is assembled for the first time. Also a series of commands are necessary to define the types of nonlinearity in the model and the combinations of loads applied to the model. Then, we run an initial job, which includes already the initial vertical load,

diana example2.dat non.com

Commands non.com

```
*FILOS
*INPUT
*ELASSE
*ELMAT
*LOADS
*ORDER
*SOLVE
*NONLIN
 SEGMENT INITIA/IN30LL
 SEGMENT EXECUT/XQ31LL
 INITIA
  ANALYS PHYSIC
  OPTION TANGEN NONSYM
  USE
     INTERF
  END USE
 END INITIA
 LOADIN
  LOAD(1): (1) 1.0 /
  LOAD(2): (2) 1.0 /
 END LOADIN
 SELECT
  NODES 2110 /
  ELEMENT NONE //
  END ELEMEN
 END SELECT
 OUTPUT TABULA NONLIN
  DISPLA TOTAL GLOBAL
 END OUTPUT
 EXECUTE LOAD(1) STEPS
  SIZE 0.5(2) /
  PERFOR NEWTON REGULA MI=15
  NORM ENERGY NEWREF CONTIN CO=1.d-6 AB=1.d+4
 END EXECUTE STEPS
*POST/CI30LL
 SEGMENT WRITE/WR30LL
 LAYOUT
  MODEL.E LI=..
  ELEMEN.
  NODES. DATA.E LI=-
 END LAYOUT
 OUTPUT GRAPHI NONLIN FI="d"
  TEXT "Total displacements"
  DISPLA TOTAL
 END OUTPUT
 LAYOUT
  MODEL.E LI=-
  ELEMEN. DATA.F
 END LAYOUT
 OUTPUT GRAPHI NONLIN FI="s"
  TEXT "Principal stresses"
  STRESS TOTAL PRINCI XX YY INTPNT
 END OUTPUT
 LAYOUT
  MODEL.E LI=-
  ELEMEN. DATA.F6
 END LAYOUT
 OUTPUT GRAPHI NONLIN FI="u"
```

```
  TEXT "Relative displacement normal to interface"
    STRAIN FORCE N INTPNT FR=0.d0
 END OUTPUT
*END
```

At this stage, nonlinearities were already encountered. We proceed, with the nonlinear analysis under the special interface arc-length procedure, `SELECT ARCLEN INTTOT`. We execute a first small load step of 0.1 (= 0.1 kN) for `LOAD(2)` followed by five steps of 2.5. The small load step is necessary, because the structure must accommodate a completely different load. This is followed by fourteen load steps of 1.0 and one load step of 0.02 when the response is almost flat.

After each run, we run Module POST, to make plots of the deformed structure, the principal stresses and the normal opening of the joints as a representation of cracking in the joints and units.

The new steps, are performed with the run commands

`diana example1.dat nonlin1.com`

`diana example2.dat nonlin2.dat`

Commands `nonlin1.com`

```
*NONLIN/CI33LL
 SEGMENT EXECUT/XQ31LL
 SELECT
  NODES 2110 /
  ELEMENT NONE //
  END ELEMEN
 END SELECT
 OUTPUT TABULA NONLIN
  DISPLA TOTAL GLOBAL
 END OUTPUT
 SELECT ARCLEN INTTOT
 END SELECT ARCLEN
 EXECUTE LOAD(2) STEPS
  LINE SEARCH
  SIZE.P 0.1(1) 2.5(5) /
  PERFOR.R NEWTON REGULA MI=15
  NORM ENERGY NEWREF CONTIN CO=1.d-6 AB=1.d+4
 END EXECUTE STEPS
*POST
:    like nonlin1.com
```

28 lines skipped

Commands `nonlin2.com`

```
*NONLIN/CI33LL
```

13 lines skipped

```
SIZE.P 1.0(14) 0.02(1) /
```

3 lines skipped

```
*POST
:    like nonlin1.com
```

28 lines skipped

After each load step, the selected data for node `2110` is written to the output file `diana.tb` as specified in the `SELECT` block. A selection of the tabulated output of the last load step is given below.

Nonlinear output nonlin2.com

```
 Analysis type       : NONLIN
 Step nr.            :    23
 Load factor         :   0.2567E+02
 Result              : DISPLA TOTAL  TRANSL
 Axes                : GLOBAL


    Nodes          TDtX         TDtY         TDtZ
     2110   -0.1218E+01  0.8588E-01  0.0000E+00
```

5.3 Interpretation of the results

From the tabulated output, it is possible to obtain a force-displacement diagram with the `scan` filter. The run command is

`scan < diana.tb >> xy.dat`

Filter scan

```
awk 'BEGIN { displa_line=10; load_line=0;
             f = "%15.4e %15.4e\n"
             printf f, 0.0, 0.0
           }
/TDtY/     { displa_line = 0 }
/Load/  { load_line = 1 }
{
displa_line += 1
if ( load_line == 1 ) { force = $4
   load_line = 0 }
if ( displa_line == 2 ) { disp = $2
   printf f, -disp, force }
}
```

The horizontal force-horizontal displacement diagram for the top of the wall is given in Figure 19. A very ductile type of failure was encountered. This is due to the dry friction between the rigid blocks at the ultimate stage. Nevertheless, it is noted that inelastic (hardening) behavior in the cap can be observed in the compressed toes. The other figures show the plots produced after steps 2, 8 and 23. For each load step, we plot the deformed mesh, plotted with a solid line, the principal stresses at the integration points, plotted with a solid and dashed line, and the normal opening of all interface elements, plotted with a variable thickness solid and dashed line.

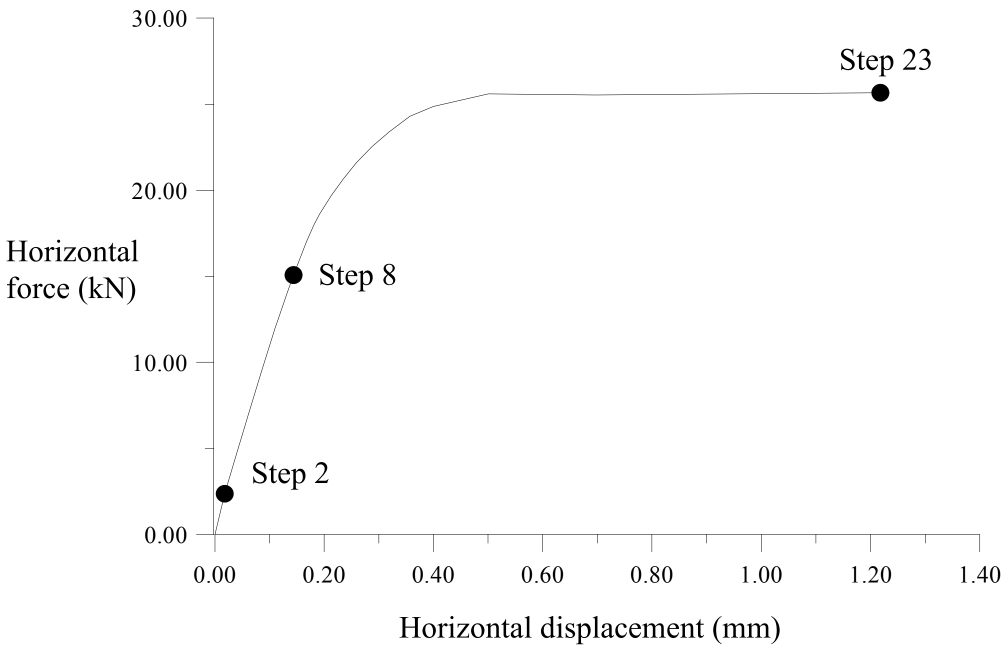
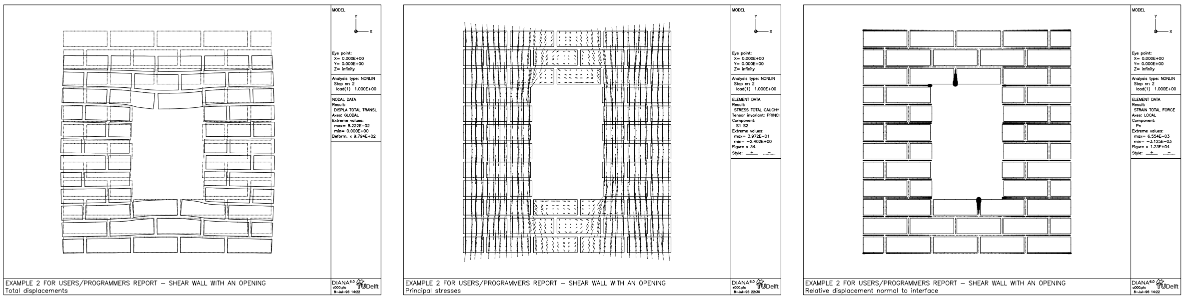
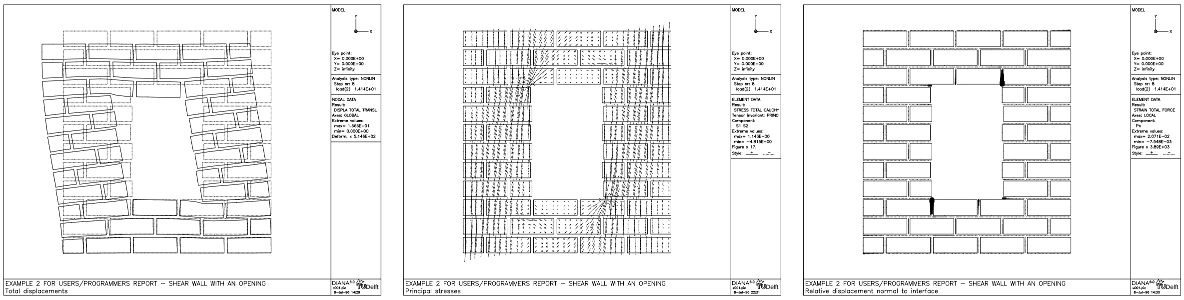


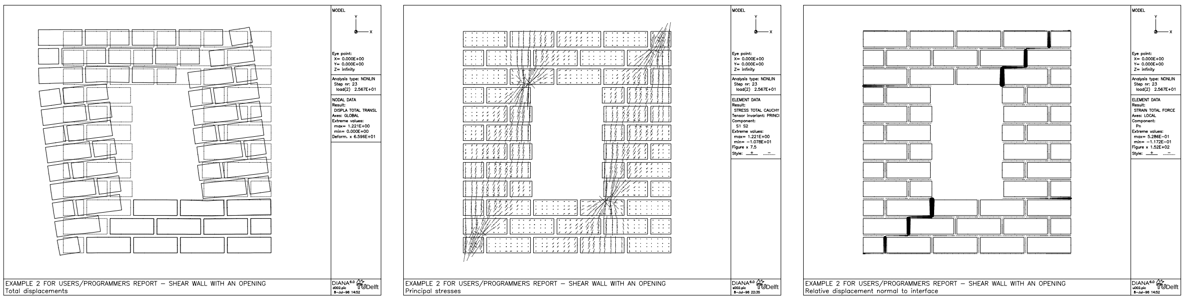
Figure 19 - Force-displacement diagram for masonry shear wall with an opening



Step 2



Step 8



Step 23

Figure 20 - Plots for masonry pier with point load: incremental deformed mesh, principal stresses and relative normal opening of interfaces

Bibliography

ATKINSON, R.H., AMADEI, B.P., SAEB, S. and STURE, S. (1989) - Response of masonry bed joints in direct shear. *J. Struc. Engrg.*, ASCE, **115(9)**, p. 2276-2296.

CEB (1991) - *CEB-FIP Model code 1990*. Bulletin D'Information no. **203**, **204** and **205**, Comite Euro-International du Beton, Lausanne, Switzerland.

CEN (1995) - Eurocode 6: Design of masonry structures. ENV 1996-1-1:1995, CEN, Brussels, Belgium.

CUR (1994) - *Structural masonry: an experimental/numerical basis for practical design rules* (in Dutch). Report **171**, CUR, Gouda, The Netherlands.

LOURENÇO, P.B. (1996) - *Computational strategies for masonry structures*, Dissertation, Delft University of Technology, Delft, The Netherlands.

SCHELLEKENS, J.C.J. (1992) - *Computational strategies for composite structures*. Dissertation, Delft University of Technology, Delft, The Netherlands.