Shear Failure Analysis of a Masonry Wall
1 Description

In this tutorial a two-dimensional masonry wall with window opening is loaded in shear until failure. Both geometric and material nonlinear effects are considered. Different modelling approaches, such as using the smeared engineering masonry model and the discrete masonry modelling with interfaces for bed- and head-joints are demonstrated. The case study describes how the model can be defined with different material models and the corresponding results.

The geometry of the masonry wall considered in this tutorial is shown in Figure 1, while the dimensions of the brick unit is shown in Figure 2.
The material properties for the masonry bricks are reported in the following tables. The material models used in the example are the linear-elastic model [Table 1], the engineering masonry model [Table 2] and the total strain crack model [Table 3]. The properties for the concrete beam and lintel are listed in Table 4.

Table 1: Material properties - Brick linear

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus $E$</td>
<td>1.74E+10  N/m²</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.15</td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>1700 kg/m³</td>
</tr>
</tbody>
</table>

Table 2: Material properties - Engineering masonry

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young modulus $E_x$</td>
<td>4E+9  N/m²</td>
</tr>
<tr>
<td>Young modulus $E_y$</td>
<td>6E+9  N/m²</td>
</tr>
<tr>
<td>Shear modulus $G_{xy}$</td>
<td>2E+9  N/m²</td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>1700 kg/m³</td>
</tr>
</tbody>
</table>

Cracking parameters

- Tensile strength head-joint defined by friction $f_t$ = 250000 N/m²
- Minimum tensile strength head-joints $f_{t,min}$ = 250000 N/m²
- Fracture energy in tension $G_{F1}$ = 18 N/m
- Residual tensile strength $f_{t,res}$ = 50000 N/m²
- Angle between stepped diagonal crack and bed-joint $\alpha$ = 0.436332 rad

Crushing parameters

- Compressive strength $f_c$ = 8.5E+6 N/m²
- Fracture energy in compression $G_c$ = 15000 N/m
- Factor to strain at compressive strength $\eta$ = 2
- Unloading factor $\lambda$ = 1

Shear failure parameters

- Friction angle $\phi$ = 0.642501 rad
- Cohesion $c$ = 350000 N/m²
- Fracture energy in shear $G_{sh}$ = 250 N/m

Table 3: Material properties - Brick nonlinear

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young modulus $E$</td>
<td>1.74E+10  N/m²</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.15</td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>1700 kg/m³</td>
</tr>
</tbody>
</table>

Tensile behaviour

- Tensile strength $f_t$ = 250000 N/m²
- Ultimate strain $\varepsilon_u$ = 18
- Residual tensile strength $f_{t,res}$ = 50000 N/m²

Compressive behaviour

- Compressive strength $f_c$ = 8.5E+6 N/m²

Table 4: Material properties - Concrete

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus $E$</td>
<td>3E+10  N/m²</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.15</td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>2200 kg/m³</td>
</tr>
</tbody>
</table>
The material properties for the interface materials are listed in the following tables. Different approaches are followed in order to describe the interface behaviour, including the use of a linear-elastic model [Table 5], the combined cracking-shearing-crushing plastic material model [Table 6], the Coulomb friction model [Table 7] and the no-tension nonlinear elastic model [Table 8].

### Table 5: Material properties - Linear Interface

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal stiffness $k_n$</td>
<td>1.0E+12 N/m^3</td>
</tr>
<tr>
<td>Shear stiffness $k_s$</td>
<td>1.0E+12 N/m^3</td>
</tr>
</tbody>
</table>

### Table 6: Material properties - Interface plastic model

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal stiffness $k_n$</td>
<td>8.3E+10 N/m^3</td>
</tr>
<tr>
<td>Shear stiffness $k_s$</td>
<td>3.6E+10 N/m^3</td>
</tr>
</tbody>
</table>

**Cracking parameters**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strength $f_t$</td>
<td>250000 N/m^2</td>
</tr>
<tr>
<td>Fracture energy $G_{f1}$</td>
<td>18 N/m</td>
</tr>
</tbody>
</table>

**Shearing parameters**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cohesion $c$</td>
<td>350000 N/m^2</td>
</tr>
<tr>
<td>Friction angle $\phi$</td>
<td>0.643501 rad</td>
</tr>
<tr>
<td>Dilatancy angle $\psi$</td>
<td>0.54042 rad</td>
</tr>
<tr>
<td>Residual friction angle $\phi_{res}$</td>
<td>0.643501 rad</td>
</tr>
<tr>
<td>Confining normal stress $\sigma_c$</td>
<td>-1.3E+6 N/m^2</td>
</tr>
<tr>
<td>Exponential degradation coefficient</td>
<td>5</td>
</tr>
<tr>
<td>Parameter $a$</td>
<td>0 m</td>
</tr>
<tr>
<td>Parameter $b$</td>
<td>130 N/m</td>
</tr>
</tbody>
</table>

**Crushing parameters**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compressive strength $f_c$</td>
<td>8.5E+6 N/m^2</td>
</tr>
<tr>
<td>Factor $C_f$</td>
<td>0</td>
</tr>
<tr>
<td>Fracture energy in compression $G_c$</td>
<td>15000 N/m</td>
</tr>
<tr>
<td>Equivalent plastic relative displacement</td>
<td>0.93 m</td>
</tr>
</tbody>
</table>

### Table 7: Material properties - Interface friction

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal stiffness $k_n$</td>
<td>8.3E+10 N/m^3</td>
</tr>
<tr>
<td>Shear stiffness $k_s$</td>
<td>3.6E+10 N/m^3</td>
</tr>
</tbody>
</table>

**Friction behaviour**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cohesion $c$</td>
<td>350000 N/m^2</td>
</tr>
<tr>
<td>Friction angle $\phi$</td>
<td>0.643501 rad</td>
</tr>
<tr>
<td>Dilatancy angle $\psi$</td>
<td>0.54042 rad</td>
</tr>
<tr>
<td>Tensile strength $f_t$</td>
<td>250000 N/m^2</td>
</tr>
<tr>
<td>Reduced shear stiffness $k_{s,red}$</td>
<td>3.6E+6 N/m^3</td>
</tr>
</tbody>
</table>

### Table 8: Material properties - Interface no tension

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal stiffness $k_n$</td>
<td>8.3E+10 N/m^3</td>
</tr>
<tr>
<td>Shear stiffness $k_s$</td>
<td>3.6E+10 N/m^3</td>
</tr>
</tbody>
</table>

**Nonlinear elasticity**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Critical normal interface opening for reduction</td>
<td>3E-6 m</td>
</tr>
<tr>
<td>Normal stiffness reduction factor</td>
<td>1E-5</td>
</tr>
<tr>
<td>Critical shear interface opening for reduction</td>
<td>3E-6 m</td>
</tr>
<tr>
<td>Shear stiffness reduction factor</td>
<td>1E-5</td>
</tr>
</tbody>
</table>
2 Finite Element Model

We start a new project for two-dimensional structural analysis. We use linear mesh order [Fig. 3]. We choose meter for the unit length, kilogram for mass and newton for force [Fig. 5]. The units and the directions are displayed in the reference system section of the geometry browser [Fig. 4].
2.1 Geometry

We define a quadrilateral sheet with name *Brick* and size of $0.2 \times 0.05$ m with the lower left corner located at the coordinates $[0, 0]$ m [Fig. 6]. Then we array copy the sheet *Brick* nine times with an offset of $X = 0.2$ m [Fig. 7].
Now we copy the sheet Brick one time with an offset $X = -0.1 \text{ m}$ and $Y = 0.05 \text{ m}$, so we obtain the sheet Brick 10 [Fig. 10]. Then we array copy the sheet Brick 10 ten times with an offset of $X = 0.2 \text{ m}$ [Fig. 11].
Now we copy the 21 bricks 22 times with an offset $Y = 0.1 \text{ m}$ and we change their color [Fig. 14].

**Main menu** → **Geometry** → **Modify** → **Array copy** [Fig. 14] [Fig. 15]

**Geometry browser** < Select all shapes, click on the color icon and change into color #990000 >

---

Figure 14: Array copy sheet

Figure 15: All brick layers
We define *Sheet 1* at position [0, 0] with size in $X = 2\, \text{m}$ and in $Y = 2.3\, \text{m}$ [Fig. 16]. We intersect all the bricks with *Sheet 1* [Fig. 18].

**Main menu** ➔ **Geometry** ➔ **Create** ➔ **Add polygon sheet** [Fig. 16] [Fig. 17]

**Main menu** ➔ **Geometry** ➔ **Modify** ➔ **Intersect shapes** [Fig. 18] [Fig. 19]
We define Sheet 2 at position [0.5, 0.8] m with size in $X = -1$ m and in $Y = 1$ m [Fig. 20]. We select all the bricks and subtract the Sheet 2 [Fig. 22].

Main menu ➔ Geometry ➔ Create ➔ Add polygon sheet [Fig. 20] [Fig. 21]
Main menu ➔ Geometry ➔ Modify ➔ Intersect shapes [Fig. 22] [Fig. 23]
In order to create the top beam, we unite the 21 bricks in the two top layers into a single shape [Fig. 24]. We rename this shape as Top beam and we give gray color [Fig. 25].
Finally, we create the lintel by uniting the 6 bricks above the window [Fig. 26]. We rename this shape as *Lintel* and we give gray color [Fig. 27].

**Main menu** → Geometry → Modify → Unite shapes [Fig. 26]

**Geometry browser** → Geometry → Shapes → Brick 380 → Rename → Lintel

Geometry browser < Select Lintel, click on the color icon and change into color #999999 >

Figure 26: Unite shapes

Figure 27: Creation of Lintel
2.2 Connections

We create the connections representing the bed- and head-joint interfaces of the masonry wall. Firstly, we define a vertical transformation with method 'Explode' and factor of $X = 1$, $Y = 2$ and $Z = 1$ [Fig. 28].

Figure 28: Virtual transformation

Figure 29: Exploded view
We define a connection with name **Bed joints**, where the top edges at each row of brick (including the lintel and excluding the top beam) are selected as source edges, and where all the bottom edges at each row of bricks (including the lintel and top beam and excluding the lowest row of bricks) are selected as target edges. We select 'Interface' connection type and 'Close' mode, and we use a structural line interface element class [Fig. 30]. At this stage, we define a material for the interface with name **Linear interface** [Fig. 31] and linear interface stiffness [Fig. 32].
We define a model thickness of 0.1 m [Fig. 33 to 34].
We check the bed-joint interface creation and switch off the virtual transformation [Fig. 36].

Main menu ➔ Geometry ➔ Modify ➔ Virtual transformation [Fig. 36]

Figure 35: Creation of bed-joint interfaces
Figure 36: Reset virtual transformation
Figure 37: Masonry wall with bed-joint interfaces
Now we define a vertical transformation with method ‘Explode’ and factor $X = 1.3$, $Y = 1$ and $Z = 1$.

We define a connection with name *Head joints*, where the left edges of all bricks (including the lintel and excluding the top beam) are selected as source edges, and where the right edges of all bricks (including the lintel and excluding the top beam) are selected as target edges. We select ‘Interface’ connection type and ‘Close’ mode, and we use a structural line interface element class. The same material and element geometry as in *Bed joints* is assigned [Fig. 38].

Finally, we check the head-joint interface creation and switch off the virtual transformation [Fig. 40 to 41].
2.3 Properties

We assign the material and geometry properties to the bricks [Fig. 42]. We define a new material named *Brick linear* [Fig. 43] with linear properties [Fig. 44].

**Main menu**  ➔  Geometry  ➔  Assign  ➔  Shape Properties  ➔  Fig. 42

**Shape Properties**  ➔  Material  ➔  Add material  ➔  Fig. 43  ➔  Edit material  ➔  Fig. 44

---

**Figure 42:** Property assignments

**Figure 43:** Add material

**Figure 44:** Edit material - linear properties
We create a new element geometry with thickness of 0.1 m [Fig. 45 to 46].
We assign material and geometry properties to the concrete lintel and top beam [Fig. 47]. We define a new material named Concrete [Fig. 48] with linear properties [Fig. 49].
2.4 Boundary Conditions

We define translational supports at the bottom edges of the bricks in $X$ and $Y$ directions [Fig. 50] and at the top edge in the $Y$ direction [Fig. 51]. In addition, we define a translation support in $X$ direction at the left top corner vertex for a prescribed deformation [Fig. 52].
2.5 Loads

We define two load cases: i) *Weight* and ii) *Deform*. For the load case *Weight* we define the deadweight load [Fig. 54]. For the load case *Deform* we define a prescribed deformation of 0.001 m in the X direction at the left corner vertex [Fig. 55].

**Main menu** → Geometry → Assign → Add global loads 💎 [Fig. 54]  
**Main menu** → Geometry → Assign → Add loads 👇 [Fig. 55] [Fig. 56]  

![Figure 54: Define deadweight load](image1.png)  
![Figure 55: Attach prescribed deformation load](image2.png)  
![Figure 56: Load in masonry wall](image3.png)
2.6 Mesh

We define for all the shapes a desired element size of 0.05 m and generate the mesh [Fig. 57].

---

**Main menu** ➔ Geometry ➔ Assign ➔ Mesh properties [Fig. 57]  
**Main menu** ➔ Geometry ➔ Generate mesh [Fig. 58]

---

**Figure 57:** Set mesh properties  
**Figure 58:** Mesh  
**Figure 59:** Check connections
3 Analysis 1: Linear Material Model

3.1 Commands
We perform a nonlinear static analysis. The analysis procedure is defined by selecting a structural nonlinear analysis type [Fig. 62].

![Analysis browser](image1)

![Command menu](image2)

![Analysis browser](image3)
In the analysis type definition we activate both physically and geometrically nonlinear effects [Fig. 64].

**Analysis browser →** Linear material behavior → Structural nonlinear → Nonlinear effects → Edit properties

[Fig. 63] [Fig. 64]

---

**Figure 63:** Analysis browser

**Figure 64:** Nonlinear effects
In the structural nonlinear analysis, we first remove the default execute block. Then we define an execute block of type start for the stress initialization with name \textit{Weight load} [Fig. 65]. In this execute block, the weight load and initial stress corresponding to a linear analysis of the weight load are applied to the model [Fig. 66]. A maximum of 30 iterations per step and secant iteration scheme with line search option are selected [Fig. 67].
The displacements due to the application of this load are cleared [Fig. 69].

Figure 68: Analysis browser

Figure 69: Physic nonlinear options
We define a second execute block for load steps named *Shear load* [Fig. 70] where the deformation loading is applied in 250 steps with load factor of 1 and where the arc length control is activated [Fig. 71].

---

**Analysis browser** ➔ Linear material behavior ➔ Structural nonlinear ➔ **Execute steps - Load steps** [Fig. 70]

**Analysis browser** ➔ Linear material behavior ➔ Structural nonlinear ➔ **Shear load** ➔ Load steps ➔ **Edit properties** [Fig. 71]

**Analysis browser** ➔ Linear material behavior ➔ Structural nonlinear ➔ **Shear load** ➔ Equilibrium iteration ➔ **Edit properties** [Fig. 72]
We chose the user selection for the output items [Fig. 73 to 74]. Finally, we run the analysis.

**Analysis browser** → Linear material behavior → Structural nonlinear → Output → Edit properties [Fig. 73]

Properties - OUTPUT → Result → User selection → Modify [Fig. 74]

**Main menu** → Analysis → Run selected analysis

---

**Figure 73:** Edit output properties

**Figure 74:** Selection of results
3.2 Results

The first analysis is carried out with the linear elastic material model for the bricks and very high linear stiffness properties for the interfaces in the bed- and head-joints. In this analysis the geometric nonlinear behavior is considered.

The analysis stops at step 223 where the convergence criterion is not reached within the maximum number of iterations. The horizontal force as function of the top horizontal displacement is displayed in Figure 75. The principal strains in the deformed model in the last step are displayed in Figure 76.
4 Analysis 2: Engineering Masonry Model

4.1 Commands

For the second case we change the material model for the bricks to the engineering masonry model and keep the very high linear stiffness properties for the interfaces in the bed- and head-joints. Also in this analysis the geometric nonlinear behavior is considered. We rename the analysis as Engineering masonry model [Fig. 77] and we assign a new material to the bricks [Fig. 78] [Fig. 79].

![Analysis browser](image1)
![Shape Properties](image2)
![Add new material](image3)
![Edit material](image4)
We input the nonlinear material properties [Fig. 81 to 83].
4.2 Results

This analysis finishes after the prescribed 250 load steps. The horizontal force as function of the top horizontal displacement is displayed in Figure 84. We display the crack strains and principal stresses in the deformed model in the last step [Fig. 85 to 86].

Figure 84: Horizontal force vs. load factor graph

Figure 85: Crack pattern

Figure 86: In-plane stresses
5 Analysis 3: Combined Cracking-Shearing-Crushing

5.1 Commands

For the third analysis we assign the material model for the bricks again. In this case, we create and assign to the interface elements a new combined cracking-shearing-crushing material model that accounts for failure in both joints and bricks. We replace the linear interface material, as assigned in Figure 30 and Figure 38 to the bed and head joints respectively, by the interface plastic model material. Also in this analysis the geometric nonlinear behavior is considered.

We rename the analysis as Combined cracking-shearing-crushing [Fig. 87] and we assign the new material [Fig. 88] to the bed- and head-joints.

![Analysis browser](https://dianafea.com)

![Add material](https://dianafea.com)

![Edit material](https://dianafea.com)
5.2 Results

This analysis finishes after the prescribed 250 load steps, but after a load-factor of about 120, even if the convergence criteria is reached within the maximum number of iterations, the solution is presenting some oscillating values. Therefore, for comparison we show the horizontal force as function of the top horizontal displacement up to this load step [Fig. 90]. We display the interface relative displacements and principal stresses in the deformed model in the last step [Fig. 91 to 92].
6 Analysis 4: Total Strain Crack Model and Nonlinear Interface Models

6.1 Commands

For the fourth analysis we assign to the bricks a total strain crack model. Also in this analysis the geometric nonlinear behavior is considered.

We rename the analysis as *TSC and nonlinear interface models* [Fig. 93] and we assign the new material to the bricks [Fig. 94].
To the interface elements for the bed-joints we now assign a Coulomb friction model. We replace the combined plastic model as assigned in the Figure 88 by the interface friction material [Fig. 98 to 100].

![Main menu](https://dianafea.com)

**Main menu** ➔ **Geometry** ➔ **Assign** ➔ **Add connection** [Fig. 98] ➔ **Edit connections** ➔ **Material** ➔ **Add material** [Fig. 99] ➔ **Edit material** [Fig. 99] - [Fig. 100]

**Figure 98:** Add material

**Figure 99:** Edit material - Linear properties

**Figure 100:** Edit material - Friction properties
To the head-joints we assign a no-tension nonlinear elastic model. We replace the combined plastic model as defined in the Figure 88 by the interface no tension material [Fig. 101 to 103].
6.2 Results

This analysis finishes after the prescribed 250 load steps. The horizontal force as function of top horizontal displacement [Fig. 104]. We display the interface relative displacements and principal stresses in the deformed model in the last step [Fig. 105 to 106].

Figure 104: Horizontal force vs. load factor graph
Figure 105: Relative interface displacements
Figure 106: In-plane stresses
Figure 107 presents the force versus displacement curves for the three nonlinear material models are in the same graph.
Appendix A  Additional Information

Folder: Tutorials/MasonryWall

Number of elements $\approx 3100$

Keywords:
  ANALYS: geomet nonlin physic.
  CLASS: large.
  CONSTR: suppor.
  ELEMEN: interf lbif pstres qâmem struct.
  LOAD: deform weight.
  MATERI: consta coulom crack elasti engmas fricti gap harden isoto linear orthot parabo rotati shear soften totstr.
  OPTION: arclen bfgs direct lagran linese nonsym normal secant total update.
  POST: binary ndiana.
  PRE: dianai.
  RESULT: cauchy crack crkwdt displa force green princi reacti strain stress total tracti.
DIANA FEA BV
Thijsseweg 11
2629JA Delft
The Netherlands
T +31 (0) 88 34262 00

DIANA FEA BV
Vlamoven 34
6826 TN Arnhem
The Netherlands
T +31 (0) 88 34262 00

© DIANA FEA BV
Disclaimer: The aim of this technical tutorial is to illustrate various tools, modelling techniques and analysis workflows in DIANA. DIANA FEA BV does not accept any responsibility regarding the presented cases, used parameters, and presented results.