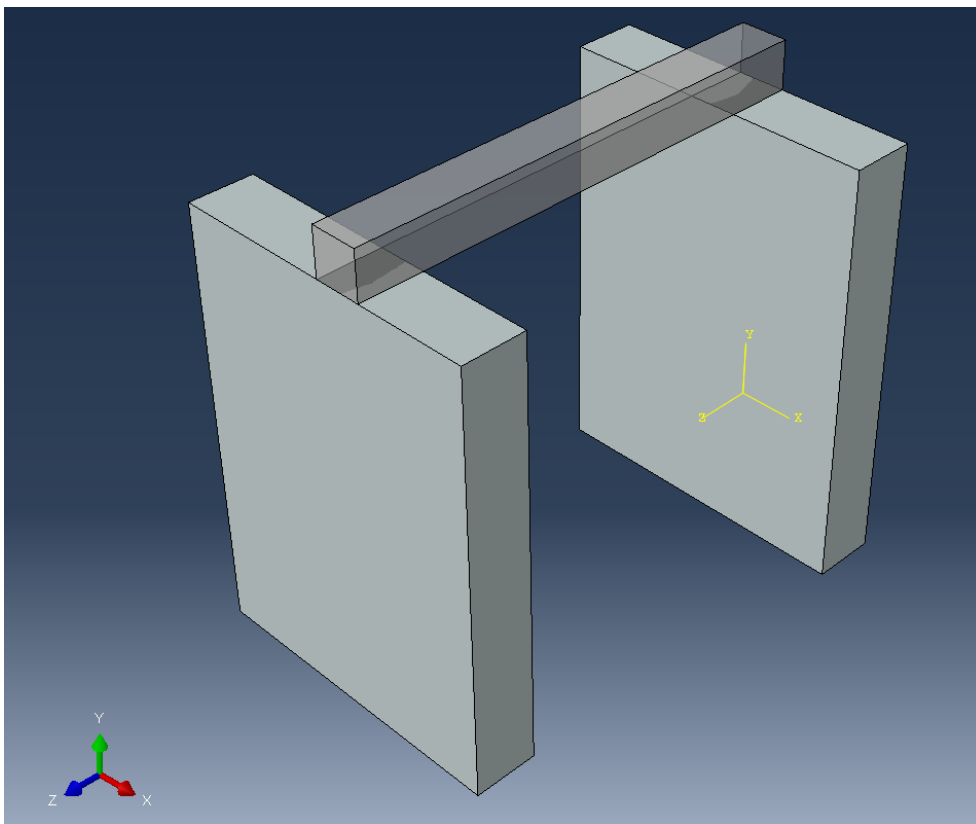


## How to model Mohr-Coulomb interaction between elements in Abaqus

### Problem description

We want to model the interaction between two masonry walls and a timber beam. The elements are connected by special interaction properties. The adopted law is Mohr Coulomb ( $\tau = c + \sigma \tan\varphi$ ) where  $\tau$  is the tangential stress,  $c$  the cohesion,  $\sigma$  the current normal stress and  $\tan\varphi$  the friction coefficient. The cohesion is assumed zero.

The walls have length 200 cm, height 270 cm and thickness of 40 cm. The timber beam has a cross section of 30x30 cm<sup>2</sup> and a length of 300 cm. Both materials are linear. The model is meshed with solid linear elements. The considered loads are (a) self-weight, (b) pressure on the beam representing live load, (c) lateral pressure on the beam simulating a push-over analysis. Moreover, a concentrated force is applied in the contact area between the beam and the wall to analyze the stress distribution over the wall.



## Geometry definition (part module)

Two parts have to be defined: walls and upper beam. They are separate parts and in the assembly they work together by means of proper interactions.

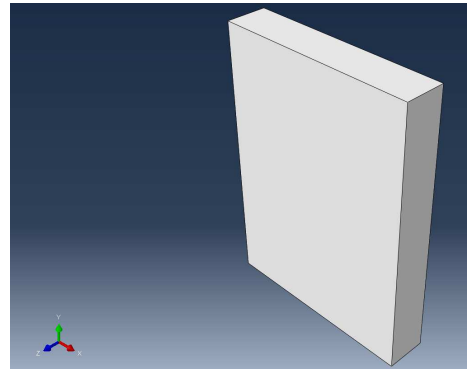
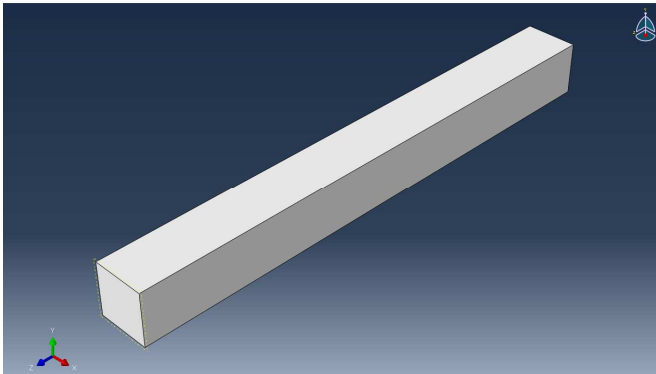


Fig. 1 - timber beam (part 1) and masonry wall (part 2)

## Material and sections definition (material-sections modules)

Masonry and timber are defined as follows:

	Specific weight (kN/m <sup>3</sup> )	Elastic modulus (MPa)	Poisson coefficient
Calcium Silicate	18.5	6900	0.15
Timber	6	10000	0.35

Remember to be consistent with units, since Abaqus does not recognize them. In the present case the force is expressed in daN and the length in cm. The sections are defined for solid homogeneous elements, for which the material has to be associated with the corresponding part.

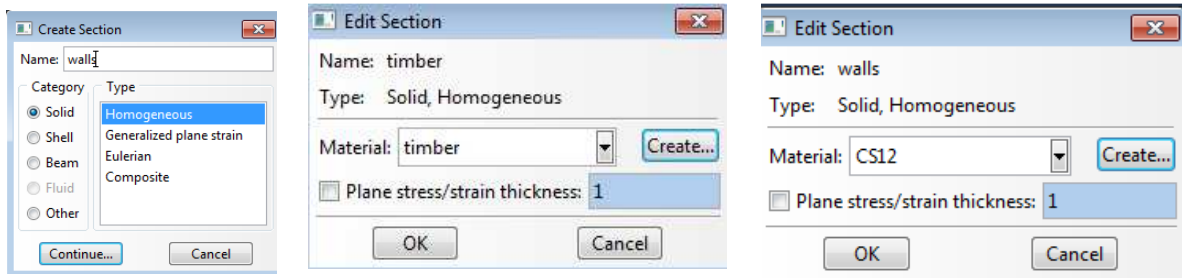


Fig. 2 - Sections definition

## Assembly module

The wall is copied with the command Instance → Linear pattern and insert in the Z direction -260. The single wall in the part module is therefore duplicated. It is suggested to follow this procedure instead of create two parts for the same wall, since the procedure is faster and cleaner.

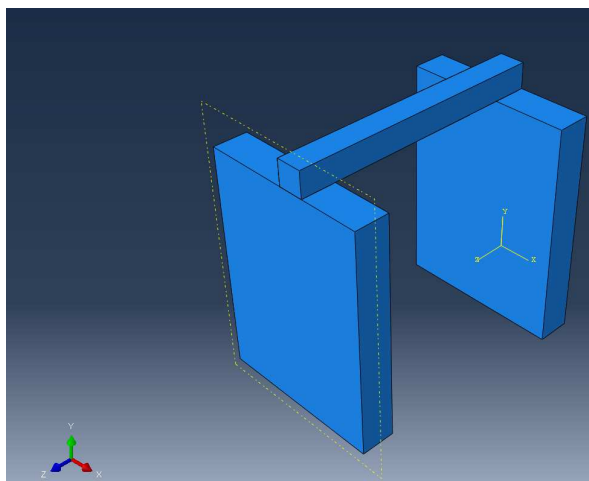


Fig. 3 - Assembly module after the duplication of the wall

At this stage, the set of base nodes and three surfaces are created.

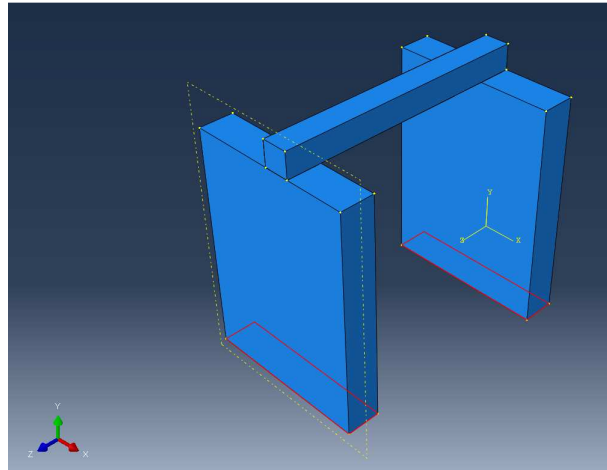
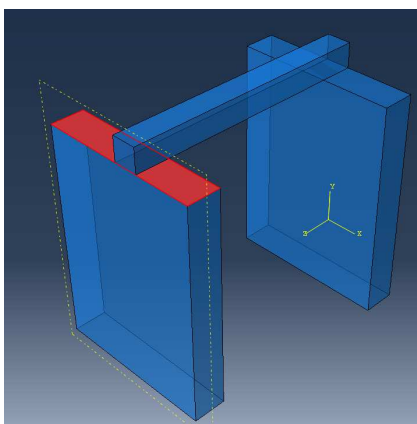
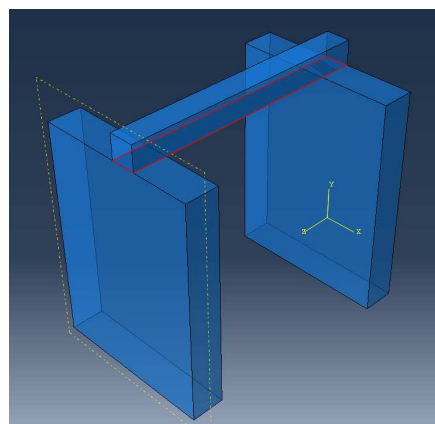


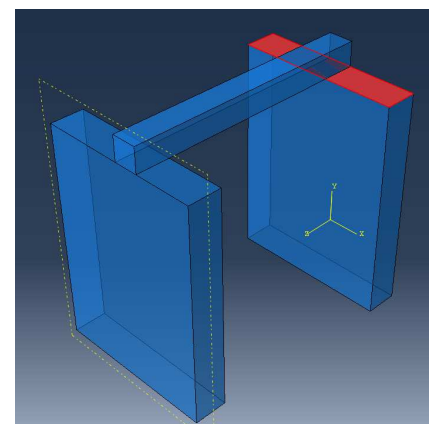
Fig. 4 - Base nodes set (geometry)



Surface 1



Surface 2



Surface 3

Fig. 5 - Surfaces

## Load steps (step module)

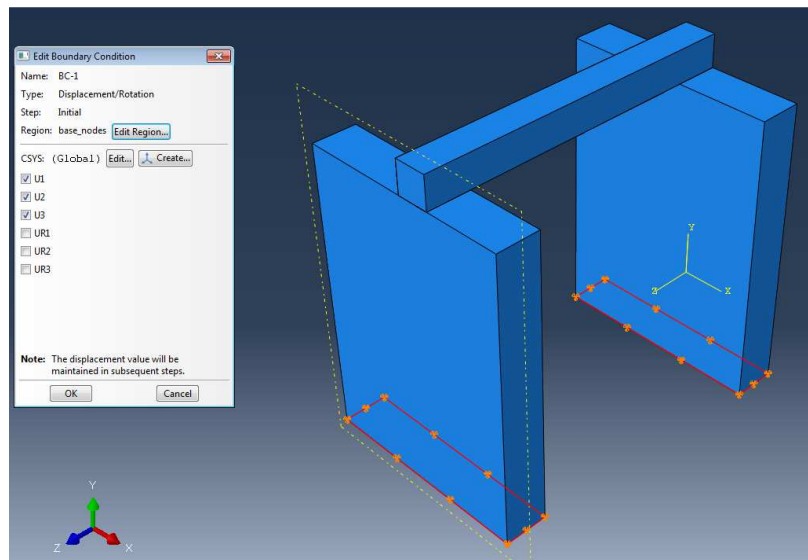


Fig. 6 - Boundary conditions in the base nodes (Initial step)

In the initial step the boundary conditions are defined. In the first step (gravity) it is necessary to define two interactions by right-clicking on Interactions→Create→Surface to Surface contact. Abaqus will automatically define the shortest distance between the nodes of the master surface (wall) and the slave surface (timber beam). However, the slave surface could be defined also in terms of nodes. Let us keep all the default options and create a contact interaction property by clicking on Create. Finite sliding is assumed. The penalty type for interactions allows some relative motion of the surfaces, whose amount

depends on slip tolerance and contact surface length. The penalty scheme, although less precise than a Lagrange multipliers approach in which the motion starts when a critical shear stress is attained, reduce the computational time.

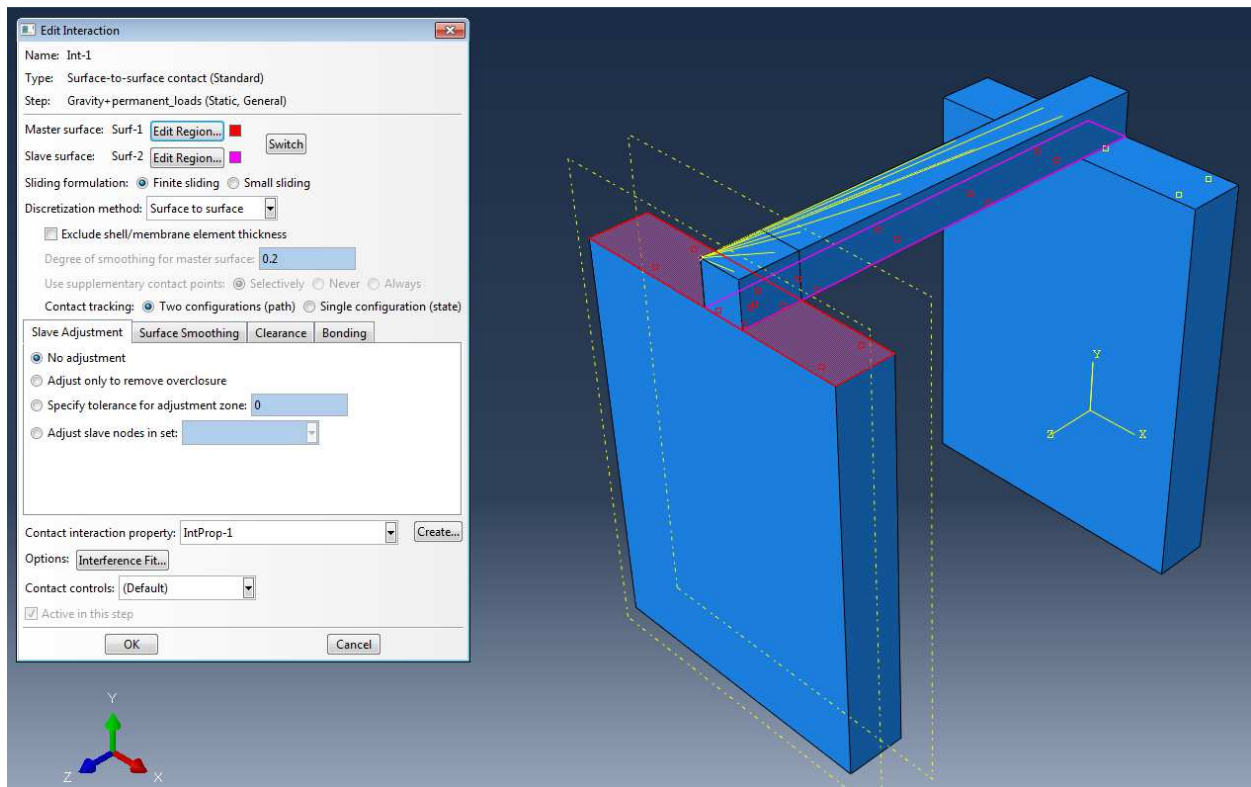


Fig. 7 - Interaction definition

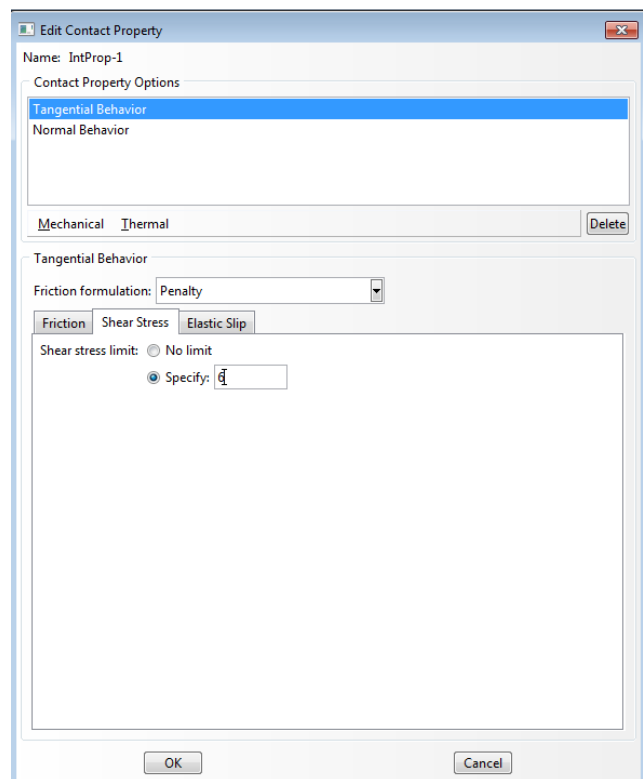
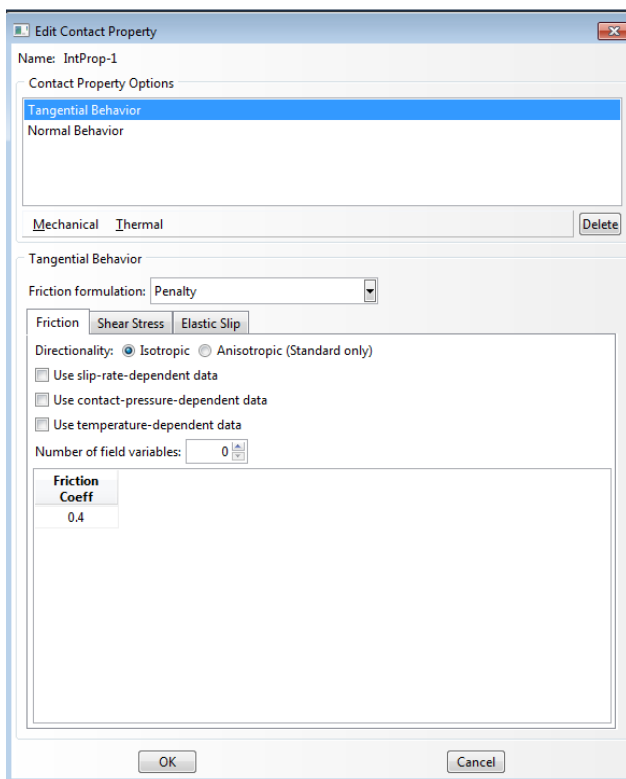


Fig. 8 - Interaction property: tangential behavior

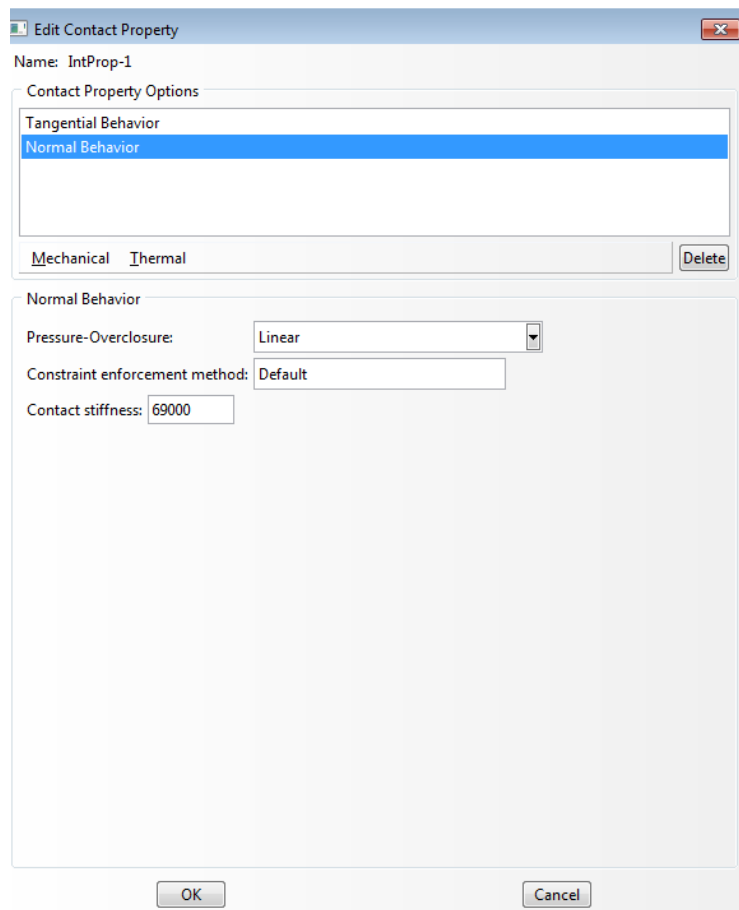


Fig. 9 - Interaction property: normal behavior

Two behaviors have to be set: tangential behavior and normal behavior. In the tangential behavior the friction coefficient is set equal to 0.4, traditional value for masonry structures. In the sub-windows Shear Stress a limit of shear stress can be defined. In our case is set equal to 6.0 daN/cm<sup>2</sup>. In the normal behavior a Linear pressure-overclosure type is defined, with a contact stiffness equal to the lower elastic modulus involved (masonry in this case, 6900 MPa). The contact behavior is modeled as a penalty contact with frictional properties. The penalty hard property allows the linear parts to deform during the overclosure with a linear response; the compressive stiffness is assumed similar to the modulus of elasticity of the weakest material, that is that of non-bearing walls (5300 MPa). The tangential stress depends upon the normal stress in each node/gauss point multiplied by the frictional coefficient.

Gravity loads are imposed with an acceleration applied to masses of 981 cm/s<sup>2</sup>.

#### Type A load condition

A live load equal to 0.4 daN/cm<sup>2</sup> is applied to the top of the beam. In a next step, a push action on the lateral side of the beam is set to 0.16 daN/cm<sup>2</sup>.

#### Type B load condition

A concentrated force of 1000 daN is applied to the contact area only in one beam extremity. It is necessary to transform this concentrated force into a pressure equal to  $\frac{1000}{30 \cdot 40} \cong 1 \text{ daN/cm}^2$ .

To simulate a rigid diaphragm it is possible to go to Constraints→Type Coupling. The control point is chosen as one of the upper beam nodes, and the slave surface is the upper surface (Fig. 11). The coupling type is kinematic and the constrained degrees of freedom are translation in the plane X-Z and rotation about Y axis. This command is actually more useful when one has slabs (rather than only one beam).

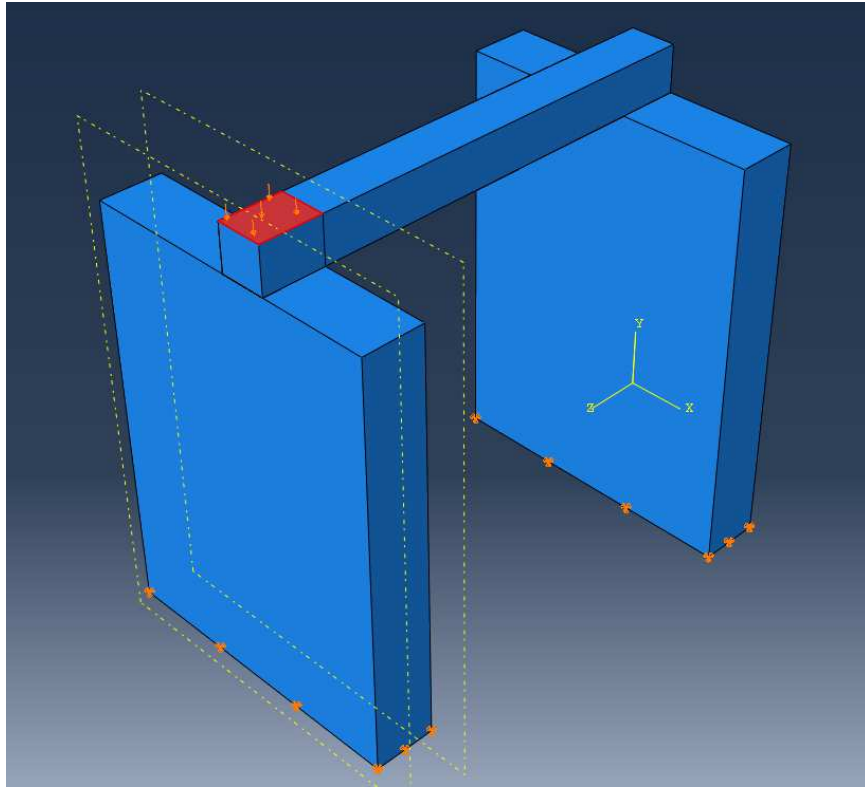


Fig. 10 - type B load

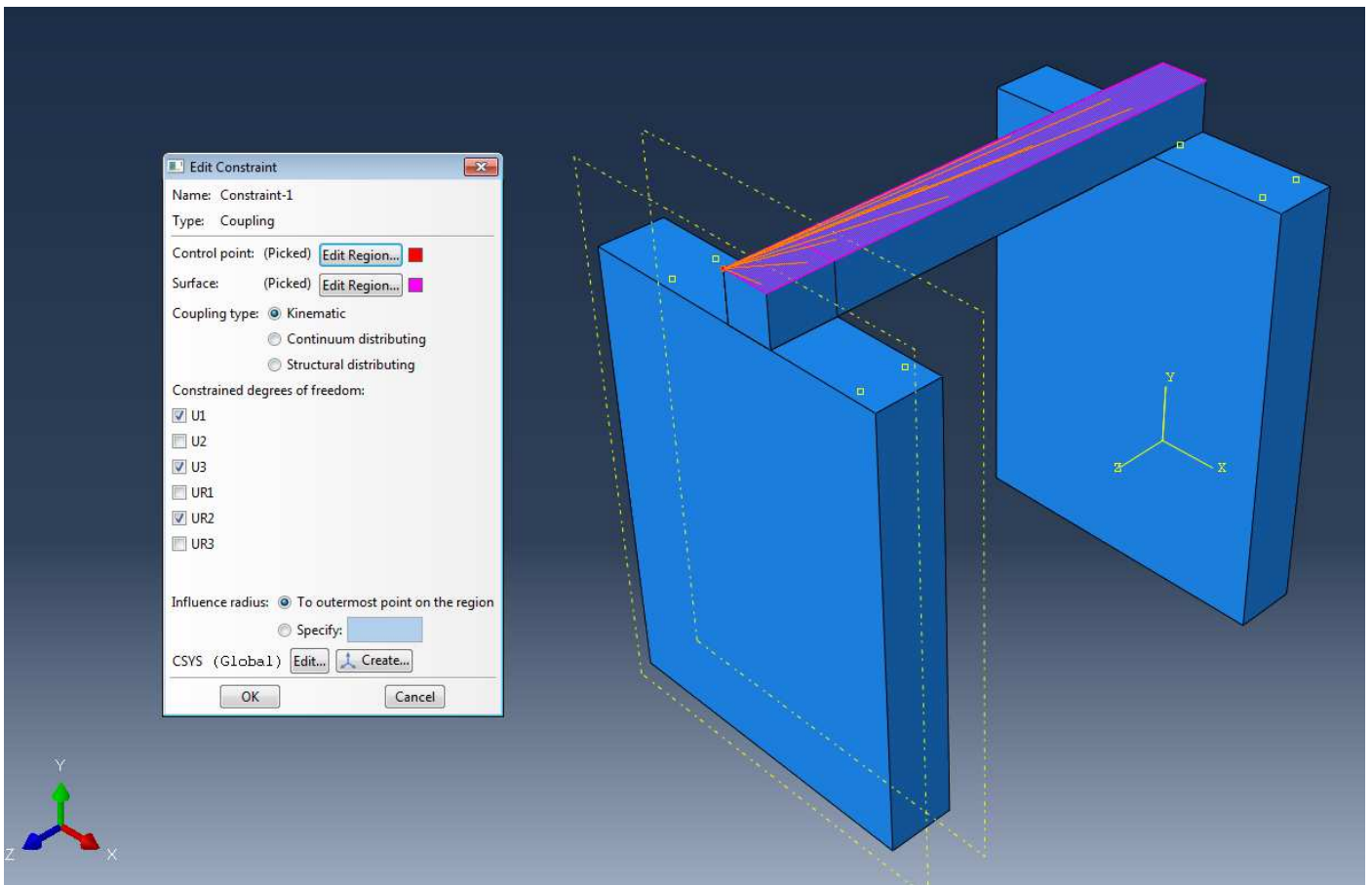


Fig. 11 - Coupling definition (rigid diaphragm)

### Mesh (mesh module)

A seed of 10 cm is defined for both parts (Seed→Part→Approximate global size). Afterwards automatically mesh the parts with Mesh→Part→OK . Keep the other defaults values also for the mesh. However, one can easily change the mesh size or the element type with quadratic elements.

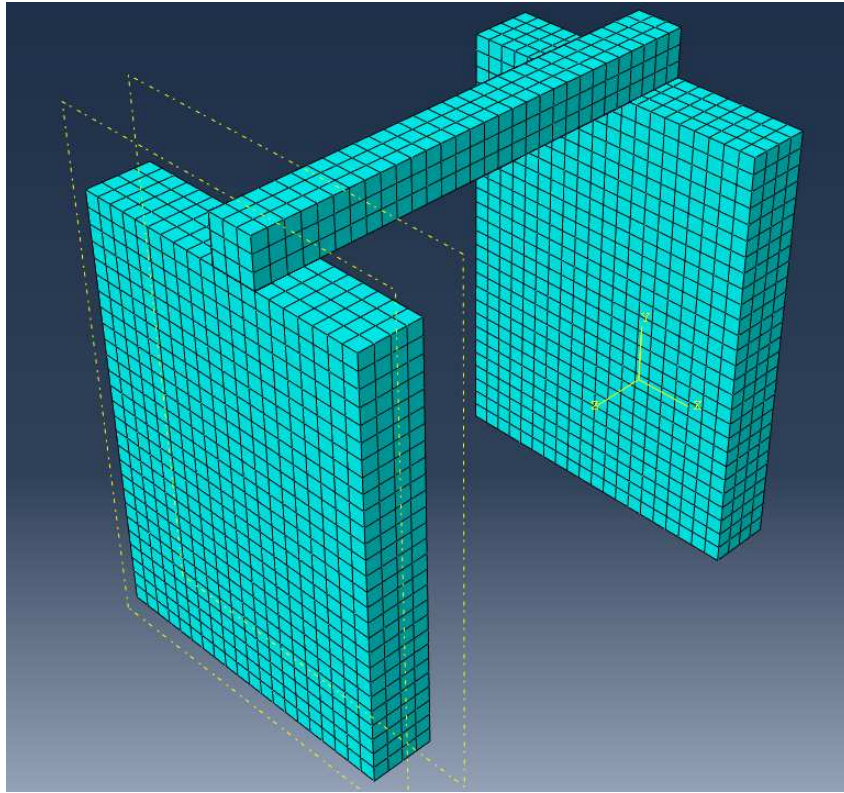


Fig. 12 - mesh with 10 cm size solid elements

### Results for Type A load condition

We plot the maximum /minimum principal stresses in the loads step. When only gravity is acting, we can see a typical deformed shape of the beam, which pushes against the walls.

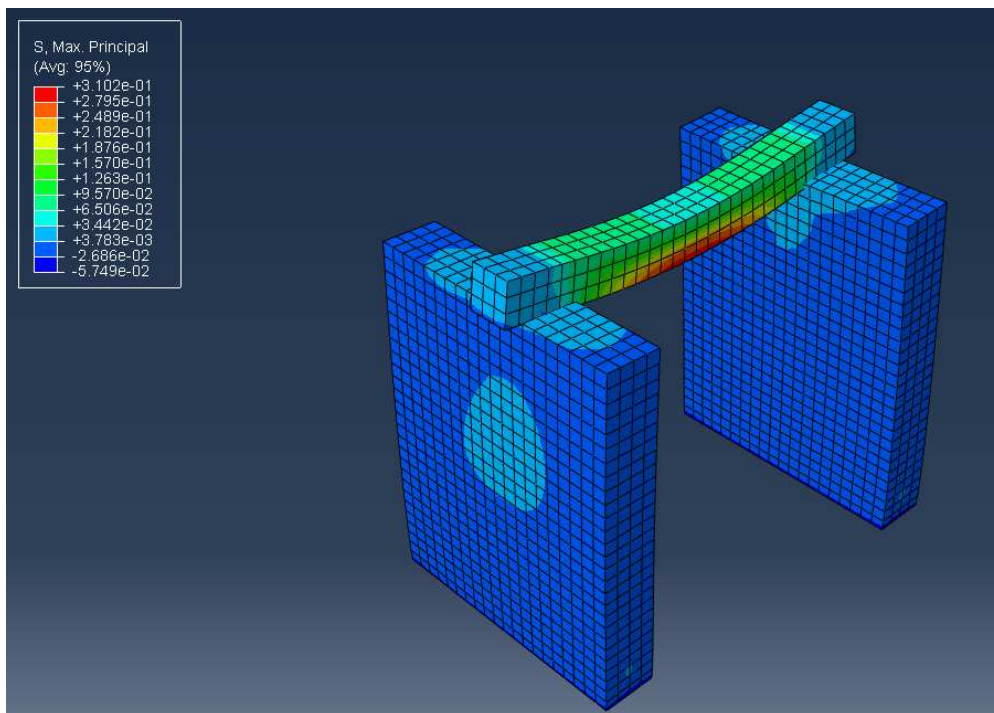


Fig. 13 - Gravity step Smax

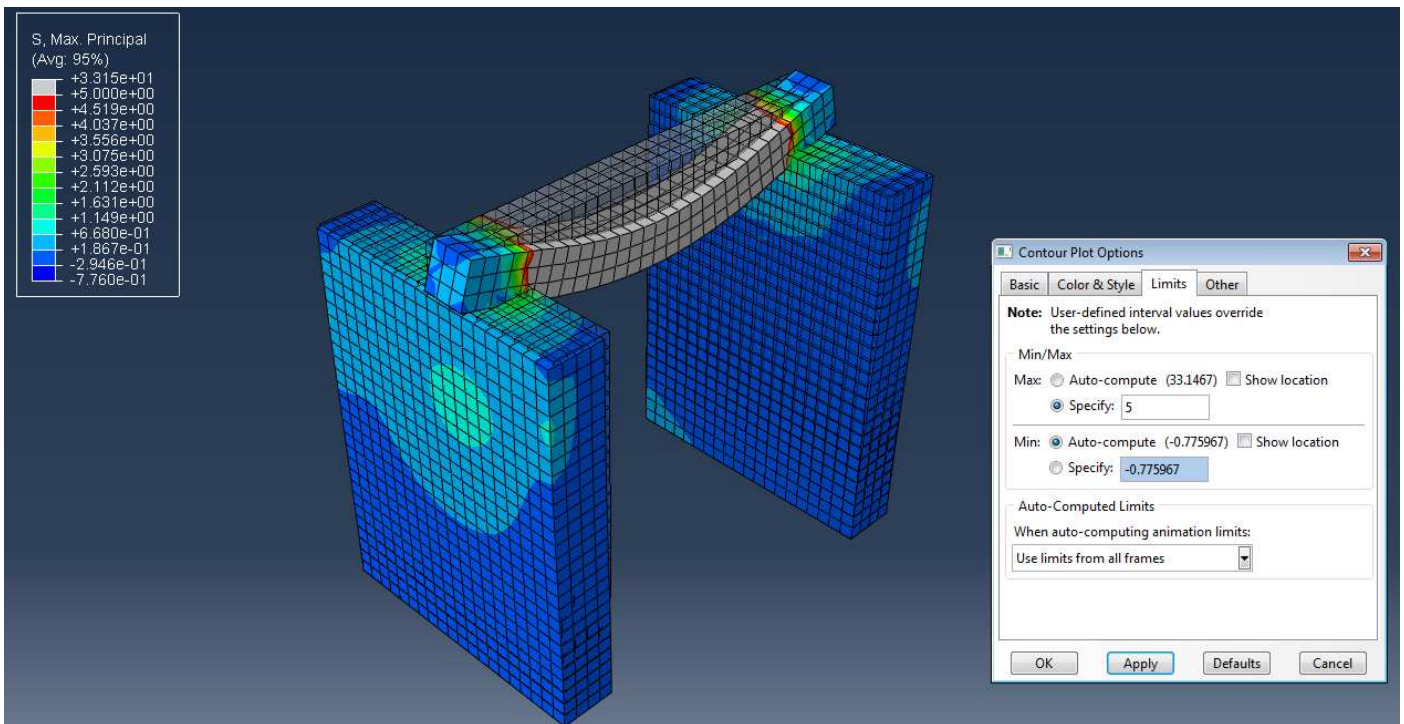


Fig. 14 - Pushover X step Smax

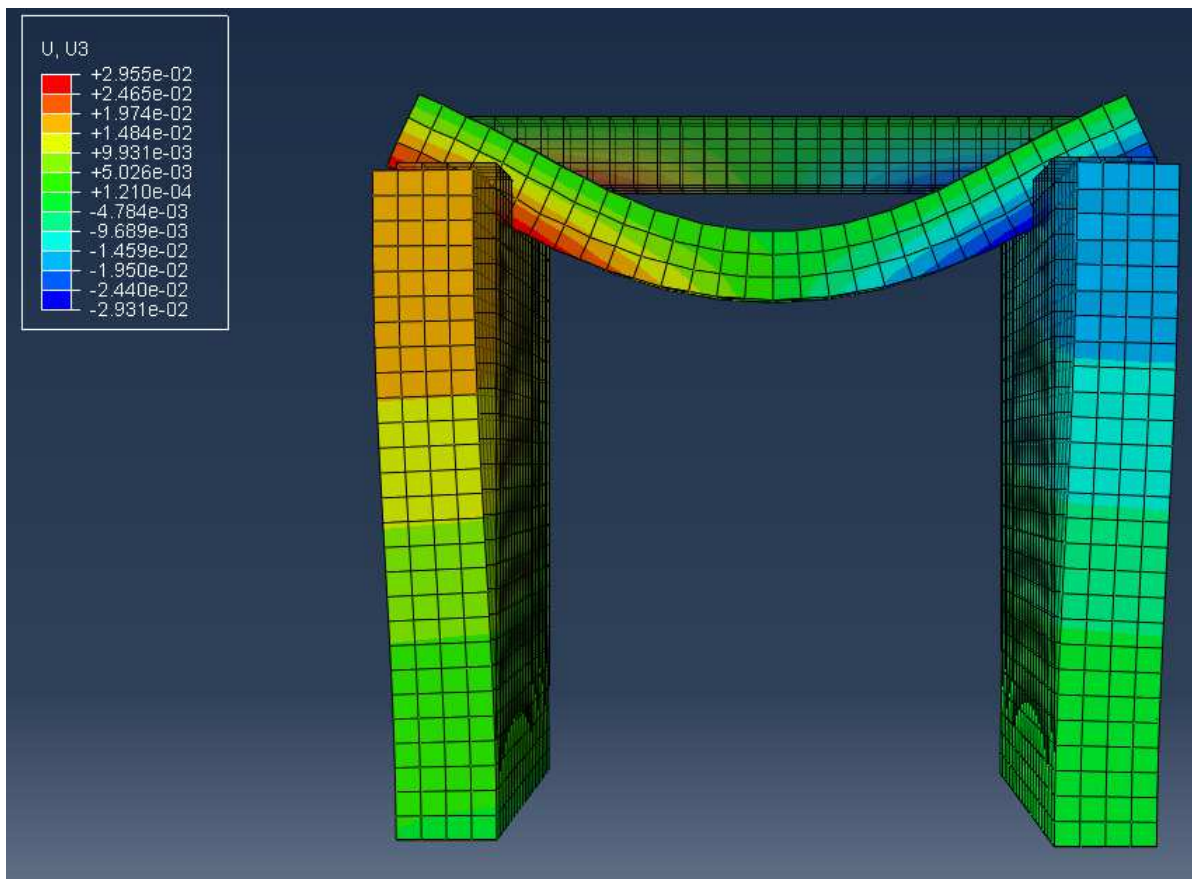


Fig. 15 - Pushover X step deformed shape in horizontal direction: opening of the walls

The calculation converged since the maximum shear strength was never attained. It is interesting to read the stress distribution over the wall, caused by the effect of the timber beam.



## Results for Type B load condition

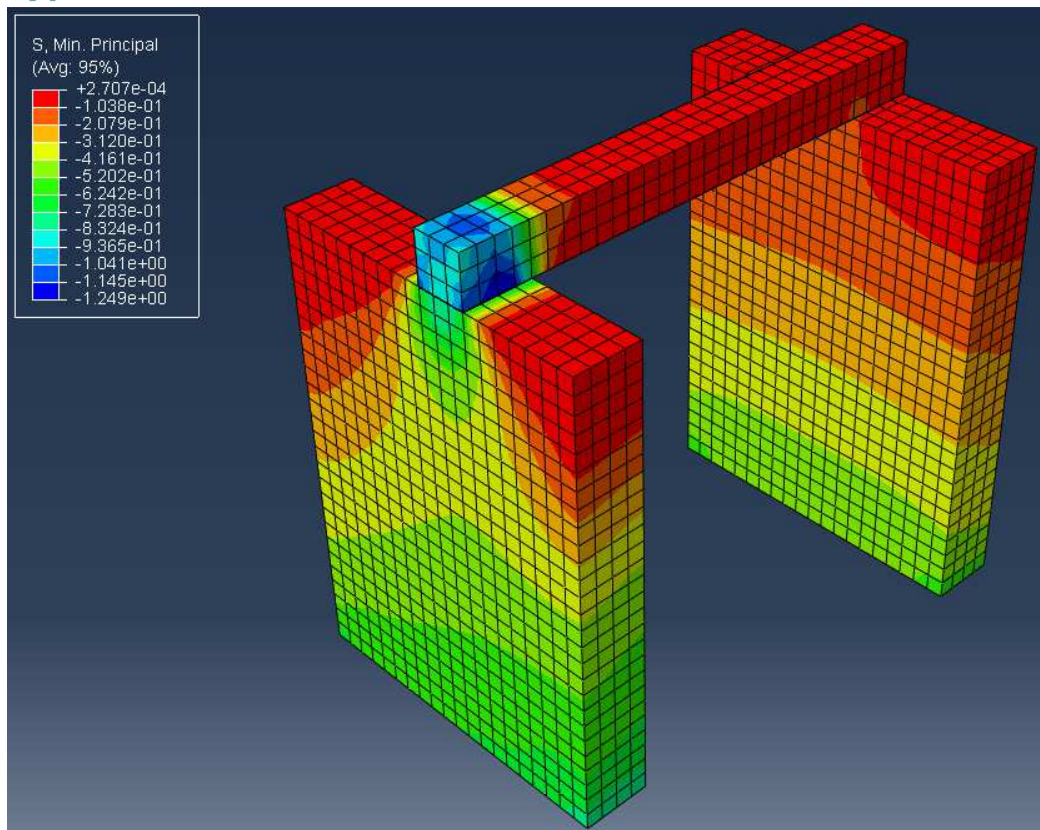


Fig. 16 - Stress distribution Smin (45°)