- Benchmark case -

Simply supported beech LVL beam with point load at midspan

1 Objectives

This benchmark aims to offer reference values for the FE-modelling of simply supported beech LVL beams with point load at midspan regarding

- the maximal vertical displacement and
- the maximal bending stress at midspan.

2 Scope

This benchmark can be used for 1st level validation of FE-models of structural systems similar to Fig. 1**.** This benchmark can be used for 2nd level validation of FE-models of

- simply supported or multi-span beams with h / $l_s \leq 4$,
- which are not imperfections-sensitive (prone to lateral torsional buckling),
- with point and/or line loads causing no relevant cross-section compressions in y-direction,
- with governing deformations due to bending in y-direction and shear in x-y-plane,
- made of solid timber according to EN 338, glulam according to EN 14080, LVL according to ETA-14/0354 or timber products with material properties in the same value range,
- within the range of linear elastic material behaviour.

3 Geometry definition, structural system and coordinate systems

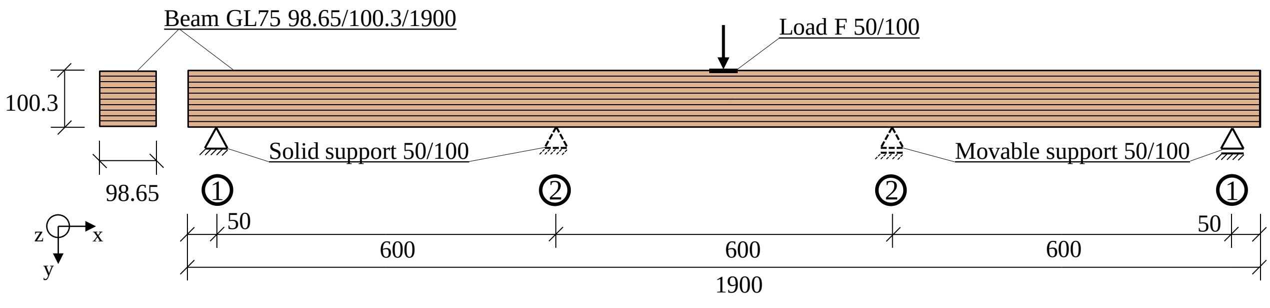

Fig. 1: Structural system with span: 1. l_s = 1.800 mm, 2. l_s = 600 mm

In this benchmark the structural system of a three-point bending test is modelled with fork bearings and two spans l_s of 600 mm and 1800 mm. The cross-section dimensions correspond to experimental investigations. The global coordinate system is shown in Fig. 1.

4 Material properties

The wood stiffness properties are defined with indices referring to the wood fibres' directions:

L = longitudinal
R = radial
T = tangential

The first index (Tab. 1) indicates the direction of the force direction, the second of the deformation.

The material properties of beech LVL were determined in bending and compression tests [1]. This research was carried out within the scope of the Cluster of Excellence 2120/1 – 390831618 / RP7.

Tab. 1: Material properties of the bending test radial/edgewise V1 A09-A11 [1]

Modulus of elasticity [N/mm²]	**Shear Modulus** [N/mm²]	**Poisson's ratio** [-]
$E_{L,m}$ = 17.566*	G_{LR} = 976	ν_{LR} = 0.2290**
		ν_{LT} = 0.6120**
E_R = 882**	G_{LT} = 1.100**	ν_{RT} = 0.1900**
		ν_{RL} = 0.0155**
E_T = 980**	G_{RT} = 50**	ν_{TR} = 0.2800**
		ν_{TL} = 0.0274**

* *Modulus of elasticity for bending with flatwise/radial loading*

** *Impact on the results of this benchmark is negliblible*

Generally the symmetry of the stiffness and the compliance matrix of timber (products) is still under discussion. In this benchmark an anisotropic stiffness matrix **C** and compliance matrix **S** according to Eq. (1) is used. The Poisson's ratios and MoE's in Tab. 1 result in unsymmetrical matrices which is implemented by means of a user defined material model (UMAT) in Abaqus.

$$\mathbf{S} = \begin{bmatrix} E_{LL} & E_{LR} & E_{LT} & 0 & 0 & 0 \\ E_{RL} & E_{RR} & E_{RT} & 0 & 0 & 0 \\ E_{TL} & E_{TR} & E_{TT} & 0 & 0 & 0 \\ 0 & 0 & 0 & G_{RT} & 0 & 0 \\ 0 & 0 & 0 & 0 & G_{LT} & 0 \\ 0 & 0 & 0 & 0 & 0 & G_{LR} \end{bmatrix} \text{ and } \mathbf{C} = \begin{bmatrix} 1/E_L & -\mu_{RL}/E_R & -\mu_{TL}/E_T & 0 & 0 & 0 \\ -\mu_{LR}/E_L & 1/E_R & -\mu_{TR}/E_T & 0 & 0 & 0 \\ -\mu_{LT}/E_L & -\mu_{RT}/E_R & 1/E_T & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G_{RT} & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G_{LT} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G_{LR} \end{bmatrix} \quad (1)$$

Alternatively symmetrical stiffness and compliance matrices may be used with ν_{LR} = 0.269, ν_{LR} = 0.552 and ν_{LR} = 0.221.

5 Loading

Point loads:

- Span 600 mm: F = 35 kN
- Span 1800 mm: F = 11 kN

The dead weight of the member is neglected as it is less than 1.0 % of the maximum load.

Abaqus Note: Modelled as concentrated force in y-direction with the load not following the rotation (see section 6 g)).

6 Boundary conditions and constraints

Boundary conditions and constraints and their application in Abaqus are shown in Fig. 2. The load is applied on the reference point “Load” and the support conditions on the reference points “Left / Right support. The deformations of the reference point “Load” have to be restrained in all but the z-direction.

Left support:

a) Displacement boundary conditions of the reference point “Supp_left_Beam” in x-, y-, and z-direction are set to zero.
b) Rotation boundary condition of the reference point “Supp_left_Beam” around the x-axis is set to zero.

→ This creates a fork bearing at the reference point “Supp_left_Beam”.

Right support:

c) Displacement boundary conditions of the reference point “Supp_left_Beam” in y-, and z-direction are set to zero.
d) Rotation boundary condition of the reference point “Supp_left_Beam” around x-axis is set to zero.

→ This creates a fork bearing at the reference point “Supp_right_Beam”.

Load support:

e) Displacement boundary conditions in x- and z-direction and rotation boundary conditions around x-, y- and z-axis of the reference point “Load” are set to zero.

Coupling of the supporting points to the beam:

f) The reference points “Supp_left_Beam” and “Supp_right_Beam” are coupled to the supported surfaces of the beam x-,y- and z- direction for transferring the support conditions.

Coupling of the load application point to the beam:

g) The reference point “Load” is coupled to the loaded surface of the beam x-,y- and z- direction for transferring the support conditions and the load.

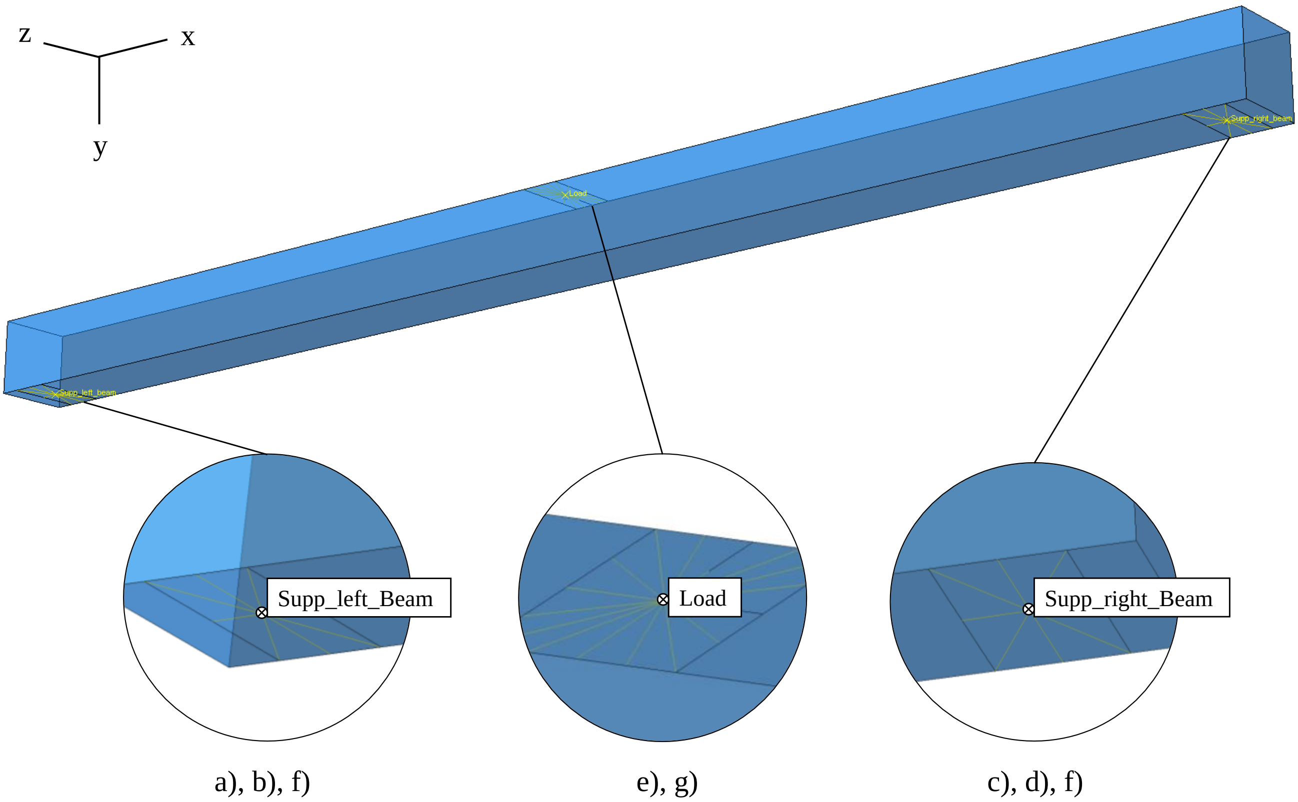

Fig. 2: Abaqus model: Boundary conditions and constraints

7 Modelling

In this benchmark one so called “part” in Abaqus is used to model the entire beam.

8 Discretization

20 node quadratic hexahedral volume elements with reduced integration (Abaqus: C3D20R) are used in this benchmark. The beam’s mesh contains 76 elements in x-direction (length), 4 elements in z-direction (width) and 20 elements in y-direction (height) (see Fig. 3). Within a verification process according to [1] it could be shown that a further mesh refinement over height and length does not yield differences in the SQR’s > 1 %. Thus further mesh refinement over the length, height and width does not lead to a notable increase in calculation accuracy.

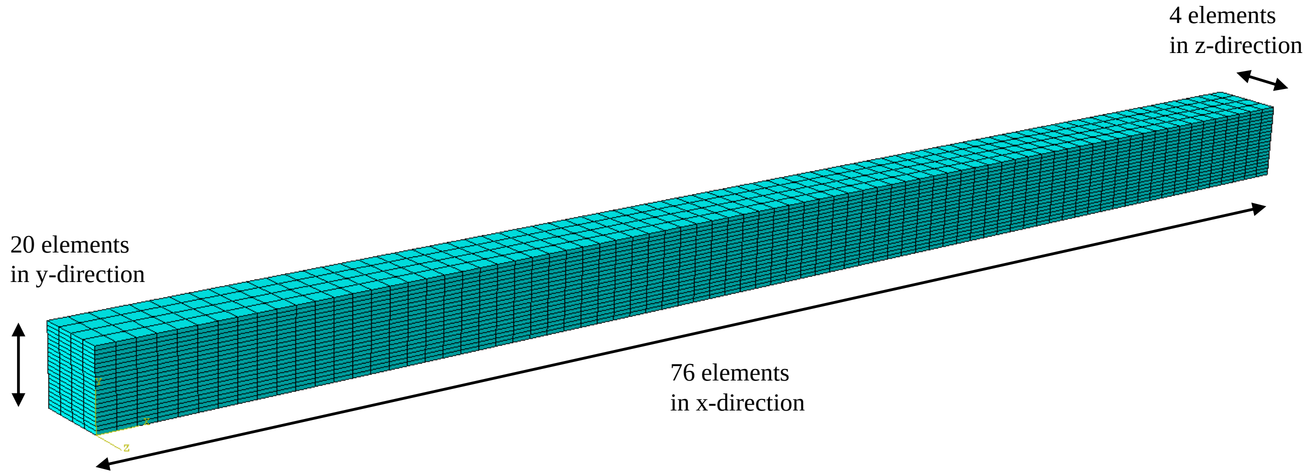

Fig. 3: Abaqus model: Discretization

9 Experimental test results

Tab. 2: Experimentally determined deformations [1].

Span l_s [mm]	**Effective modulus of elasticity** E_{eff} [N/mm²]	**Vertical deformation at midspan** w [mm]
600	10,958	1.732
1800	16,463	9.787

The effective modulus of elasticity E_{eff} can be calculated according to Eq. (2).

$$E_{eff} = \frac{F \cdot l_s^3}{48 \cdot I_y \cdot w_y} \quad (2)$$

10 Finite element analysis results

With the structural system, boundary conditions, material parameters and discretization described in the preceding chapters the results displayed in Tab. 3 can be obtained within finite element calculations. σ_x is the longitudinal stress at the lower edge of the cross-section. As this stress varies slightly over the cross-section width its values at the left/right edge and in the middle are given with $\sigma_{x,mid}$ and $\sigma_{x,edge}$. Fig. 4 to Fig. 7 display the results of the deformations and stresses of the FE-calculations.

Tab. 3: Results of Finite Element calculations

l_s [mm]	E_{eff} [N/mm²]	w [mm]	$\sigma_{x,mid}$ [N/mm²]	$\sigma_{x,edge}$ [N/mm²]
600	10,935	1.733	32.35	31.78
1800	16,416	9.805	30.13	29.86

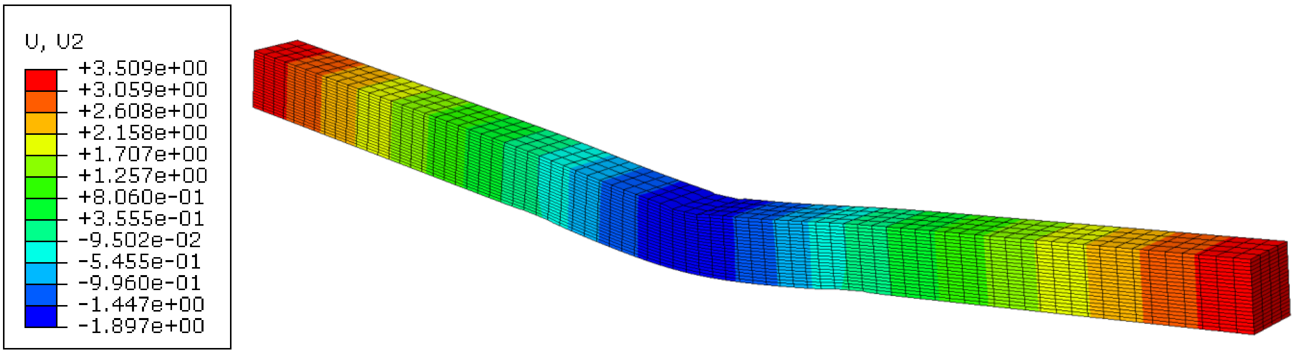

Fig. 4: Abaqus results: Vertical deformations w in y-direction in [mm] for span 600 mm

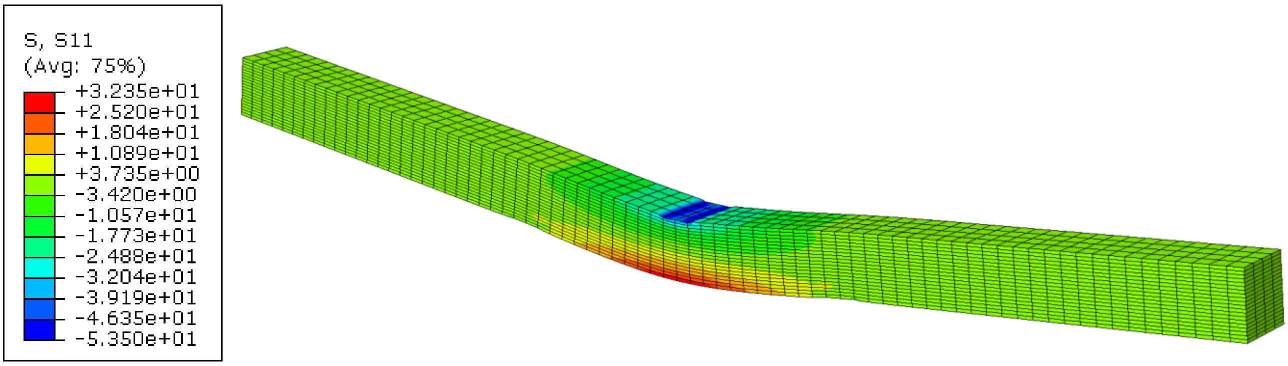

Fig. 5: Abaqus results: Bending stresses σ_x in [N/mm²] for span 600 mm

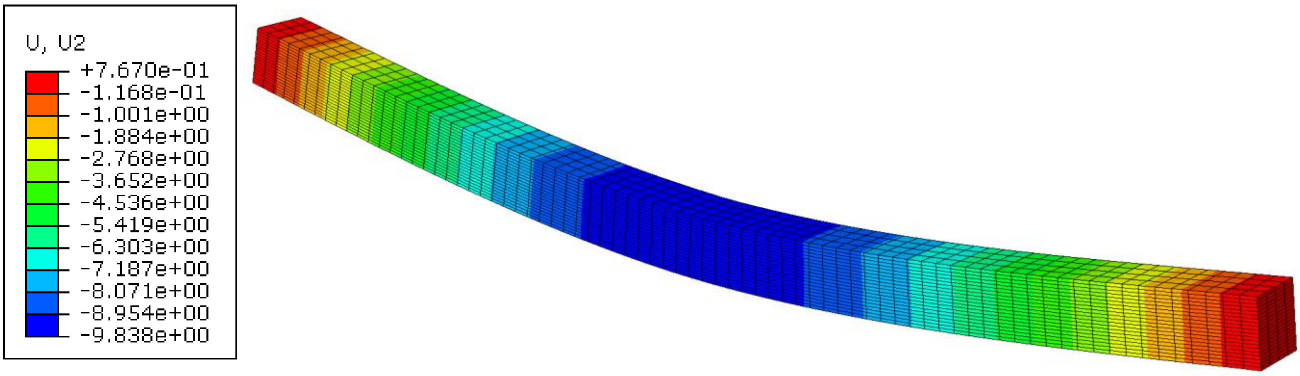

Fig. 6: Abaqus results: Vertical deformations w in y-direction in [mm] for span 1800 mm

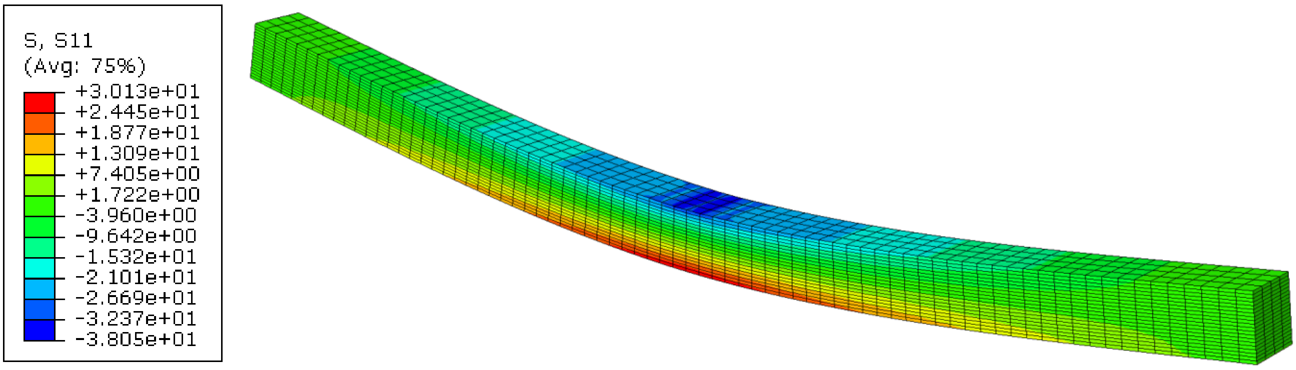

Fig. 7: Abaqus results: Bending stresses σ_x in [N/mm²] for span 1800 mm

Within the evaluation of the numerical results of the bending tests with span l_s = 600 mm, it is evident in Fig. 8 that the assumption of flat surfaces remaining flat/ Bernoulli hypothesis at midspan is no longer fulfilled. This must be taken into account when validating simply supported or multi-span beams with $h / l_s \leq 4$.

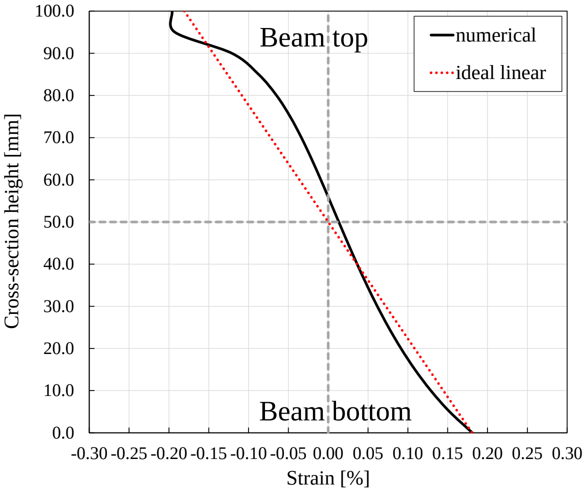

Fig. 8: Numerically determined longitudinal strains ε_x of a beam at midspan displayed over the cross-section height for span 600 mm.

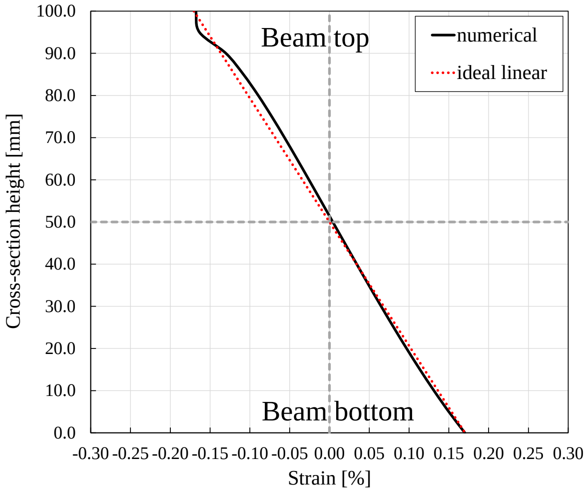

Fig. 9: Numerically determined longitudinal strains ε_x of a beam at midspan displayed over the cross-section height for span 1800 mm.

11 Comments upon difficulties that might be encountered during FE-modelling

The following section is giving hints on different kinds of general and modelling issues which might occur depending on the numerical software that is used.

- none

**Abaqus Notes:*

- Unsymmetrical stiffness matrices have to be implemented with a user defined material model (UMAT). As setting up the environment for writing such a UMAT is a bit tricky a symmetric stiffness matrix should be used in most cases.

12 References

[1] Lukas, J.: *Experimentelle und numerische Untersuchung des elastischen Materialverhaltens von Buche LVL (in German).* Bachelor Thesis, Institute of Structural Design, University of Stuttgart, Germany, April 2021.