

PLT

Plates by Finite Elements

The PLT application is intended for the analysis of plate structures of any kind according to the finite element method. The powerful graphic user interface provides numerous functions for the fast and efficient processing of this complex calculation model.

Standards

- DIN EN 1992
- ÖNORM EN 1992
- BS EN 1992
- NTC EN 1992
- EN 1992
- DIN 1045 / DIN 1045-1
- ÖNorm B 4700

PLT features

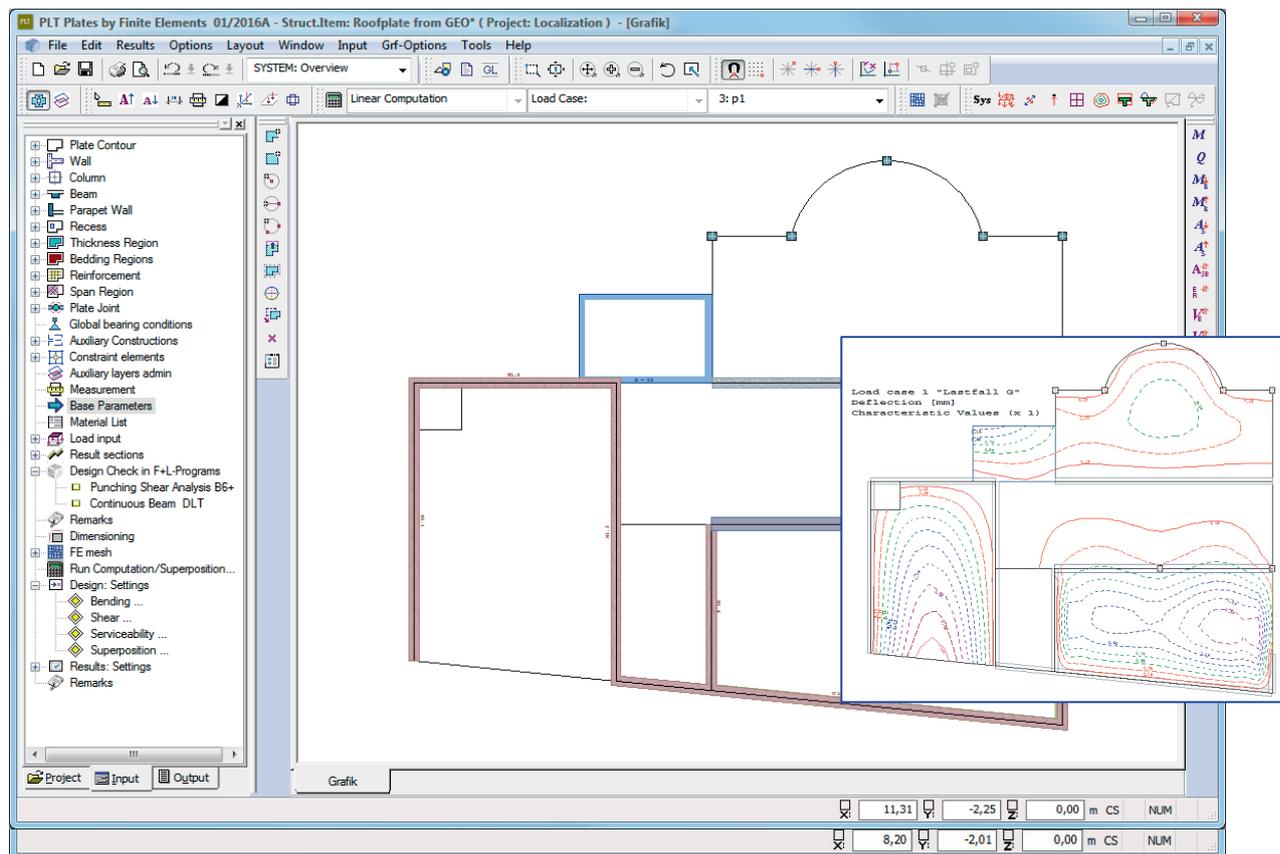
- Any outline composed of straight and curved edges can be defined
- Calculation of any plate whether susceptible to torsion or not.
- Plate joints

- Optional consideration of shear deformations of thick plates
- Calculation of the deformations in state II
- Bending- and shear design
- Crack width verification
- Automatic FE mesh generation
- Freely definable support conditions, spring rigidity can be calculated automatically
- Optional automatic recalculation of the spring rigidity of vertical components after editing (e.g. the storey height or the wall properties)
- Integrated beams with scalable bending rigidity
- Definition of any concentrated, distributed, line or temperature load
- Design situations due to earthquakes

- Comprehensive evaluation and representation options for the results, either in an output grid independent of the FE mesh, via ISO lines or along the result sections, if required

Graphic user interface

- Object-oriented input including components
- Fast input of even sophisticated outlines and unlimited editing options as well as comprehensive functions (move, copy, mirror, etc.)
- Direct import of geometric data from the CAD applications ALL-PLAN and GLASER -isb cad-
- DXF data can be used as construction aid in the background



Design / material

- Automatic design of the plate as well as the beams when using reinforced concrete
- If you do not perform a design, you can define any orthotropic material
- Data transfer to the applications DLT for the beam design and B6+ for the punching shear analysis of columns

Area definitions

- Supporting direction areas for the definition of uniaxial supporting areas
- Bedding areas for elastic bedding, you can optionally define bedding failure
- Reinforcement areas for the definition of a basic reinforcement and the setting of rotated reinforcement directions by default
- Thickness areas to describe partial plate areas with different thicknesses

Supports

Point or line supports are generated via the objects column and wall, whereby the application automatically calculates the real rigidity values, if the corresponding option was activated. There are several representation options for the bearing reactions at walls, either in the form of various diagrams in kN per linear metre or in the form of points along the wall axis.

You can optionally include tension spring failure for columns and walls in the calculation.

Loads

You can define single, line, distributed and temperature loads in any layout.

Mesh generator

The automatic mesh generator allows you to generate meshes with triangular and rectangular elements as well as mixed meshes.

FE analysis

The plate elements include displacement and stress approaches. A considerable benefit of these so-called hybrid elements is their accuracy with thin plates that are frequently used in building construction. Alternatively, elements in accordance with the Reissner-Mindlin theorem are used for thick plates. They include the shear deformations.

Superposition

The PLT application includes a fully automatic superposition feature that takes the selected regulation into account. The user can exclude particular load cases from the superposition.

Each load case is assigned to a group of actions. On the basis of this assignment, the application calculates the decisive load combination including a corresponding leading action.

You can optionally perform the calculation with alternative load cases.

T-beams

T-beams are considered by adding the rigidity terms along the beam axis. Since the plate elements do not include normal forces, the gravity axis of the beam elements is assumed to lie in the plate plane. The rigidities are calculated using Steiner portions. The beam rigidity can be increased by entering a factor.

Bending design of reinforced concrete

The design of the reinforcement is performed in accordance with the *Baumann method*. A cracked plate element is used as a model. The design approach assumes an orthogonal mesh reinforcement. The reinforcement direction is freely definable.

Shear design of reinforced concrete

The shear design is based on a bar structure model whereby the longitudinal reinforcement is taken into account. You can define by default any longitudinal reinforcement over the total plate outline to accelerate the analysis. The results reveal quickly where shear reinforcement can be avoided by increasing the longitudinal reinforcement.

Representation of the results

All results are shown at any grid point or along sections independently of the generated element mesh. The section representation also includes an overview of all results at a particular section.

You can also display the results in the form of ISO lines.

Reinforcement data transfer

Reinforcement data are transferred in an open data format referred to as ASF. The use of these data in the target system is independent of the available system functions.

FEM results transfer

The ASF format allows the transfer of all data produced in the FEM calculation. These data provide for the visualization of all results in the CAD system ALLPLAN.