

# SCN

## Panels by finite elements

The SCN application is suitable for the calculation of any load-bearing panel structure or wall-type girder in accordance with the finite elements method.

SCN uses the well-known high-performance graphic operating elements of the GEO and PLT applications. Numerous functions allow the fast and efficient handling of complex geometries.

### Standards

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

### SCN features

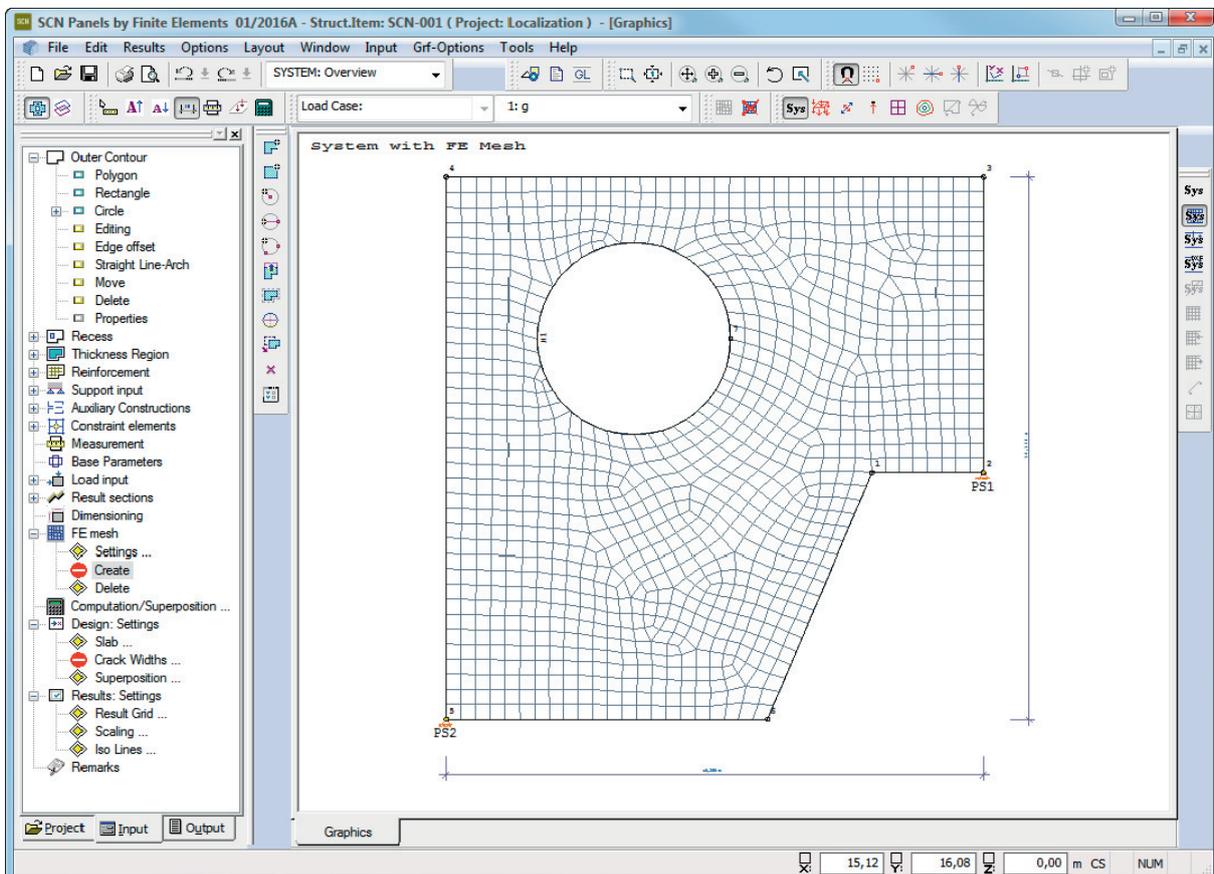
- Definition of any outline composed of straight and curved edges including block-outs
- Freely selectable bearing conditions in combination with point and line supports
- Definition of any point load or line load
- Design situations due to earthquakes
- Automatic FE mesh generation
- Comprehensive evaluation and representation options for the results, either in an output grid independent of the FE mesh, via isolines or along result sections

### Graphic user interface

- Object-oriented input including components
- Control via the menu, the main tree and the context menu
- Fast input of even sophisticated outlines and unlimited editing options and comprehensive functions e.g.
  - move,
  - copy,
  - mirror, etc.
- DXF data can be used as construction aid in the background

### Area definitions

- Reinforcement areas for the definition of a basic reinforcement and the setting of rotated reinforcement by default



- Thickness areas to describe partial panel areas with different cross sectional thicknesses

### Supports

The objects of the interactive user interface allow you to generate point or line supports directly. There are several representation options for the bearing reactions of the line supports, either in the form of various diagrams in kN per linear metre or in the form of points along the support axis.

### Loads

For the definition of loads you can select among point and trapezoidal loads in any arrangement.

### Superposition

The SCN application includes a fully automatic superposition feature that takes the selected regulation into account. The user

can exclude particular load cases from superposition.

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

You can perform the calculation using alternative load cases.

### Design

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

### Mesh generator

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

### Representation of the results

All results can be displayed at any grid point or along sections independent of the generated element mesh. Section results can be shown in a separate window as well as in the ground plan.

You can also display the results in the form of isolines.

### 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 allow the visualisation of all results in the CAD system ALLPLAN.

*Fast access to suitable menu items for the current operation via the context menu*

