

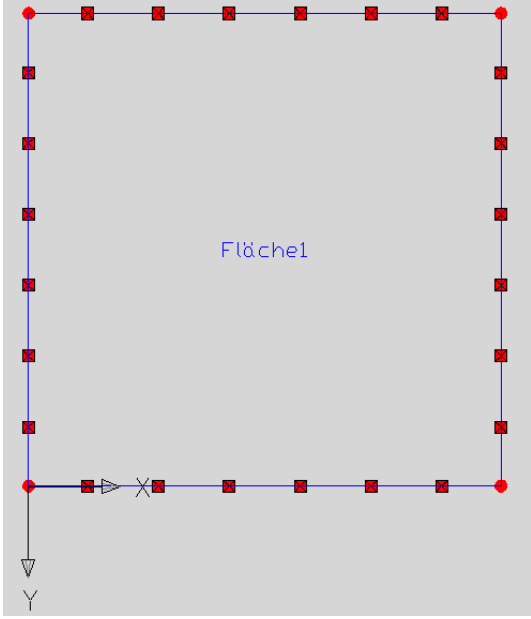
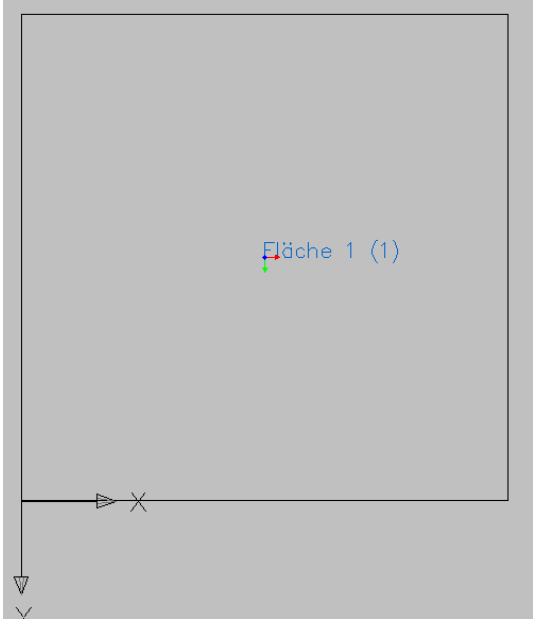
# What's new in SOFiPLUS 2010

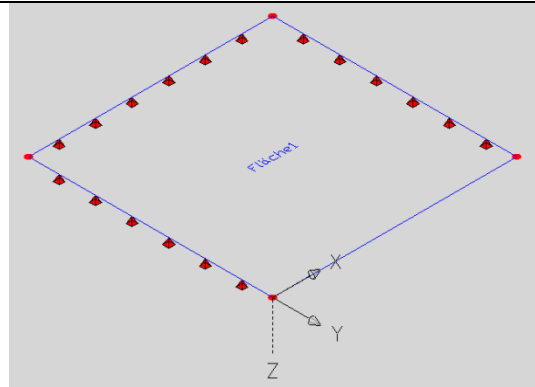
## 1 Structural Elements

The structural elements have been reworked and provide a new approach to modeling structural systems. One of the major changes is the removal of the dependencies between elements. (Previously, a line was always linked to two points and areas to several lines.)

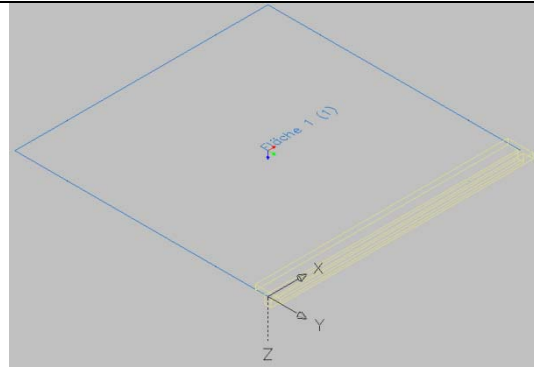
This makes modeling simpler and avoids many problems when copying, moving etc. elements. For users knowing previous incarnations of SOFiPLUS this requires a bit of change. The following points illustrate the changes:

### 1.1 Creating a slab in a 2D system

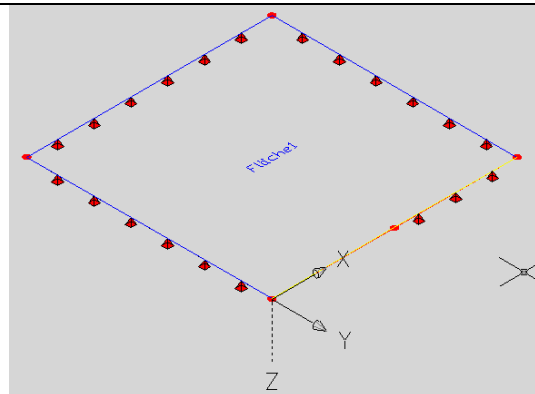
Previous Approach	New Approach
Results in 4 points, 4 lines and 1 area.	Results in 1 area.
	
To create a beam you remove the support conditions and assign a cross section to a structural line representing one of the boundary edges of the area.	To create a beam you draw a new structural line with a cross section.



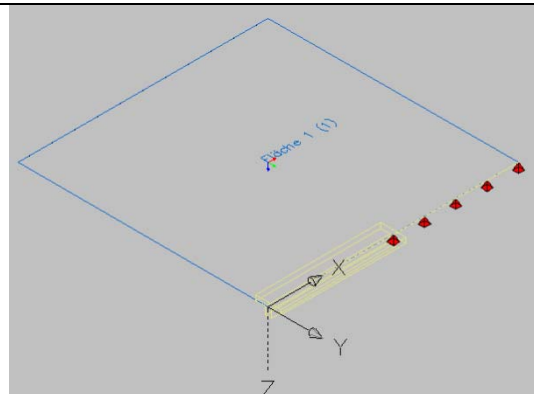
To create a beam half the length of a boundary edge and add support conditions to the other half, you split the line and remove the support conditions of and assign a cross section to one of the newly created lines.



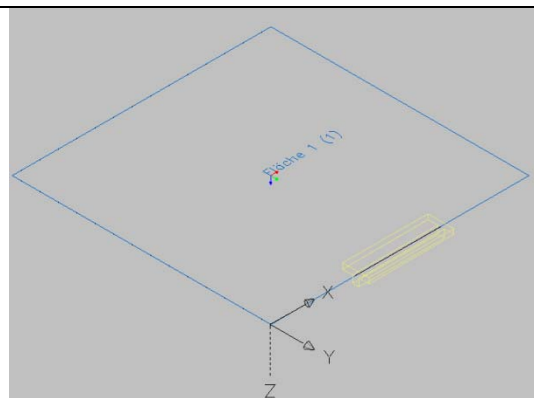
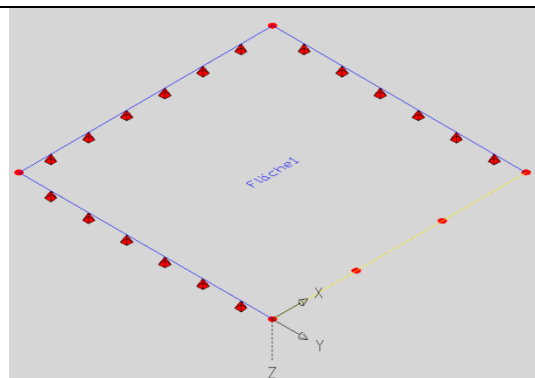
To create a beam half the length of a boundary edge and support conditions on the other half, you draw two lines, one with support conditions and one with a cross section.



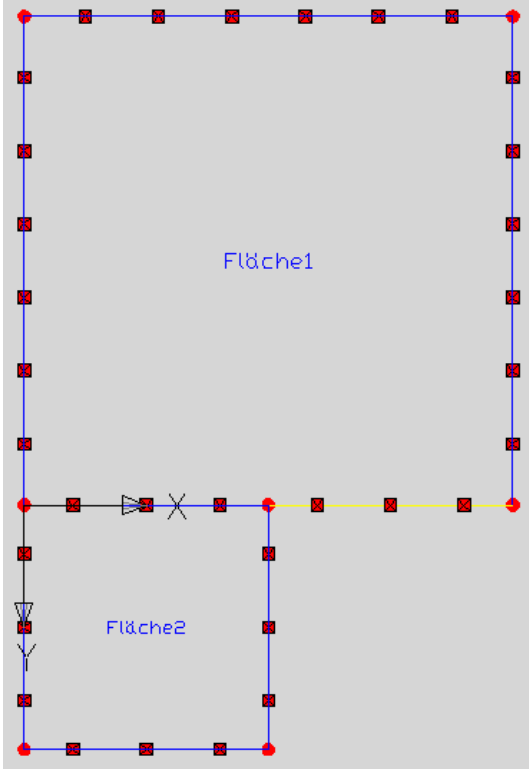
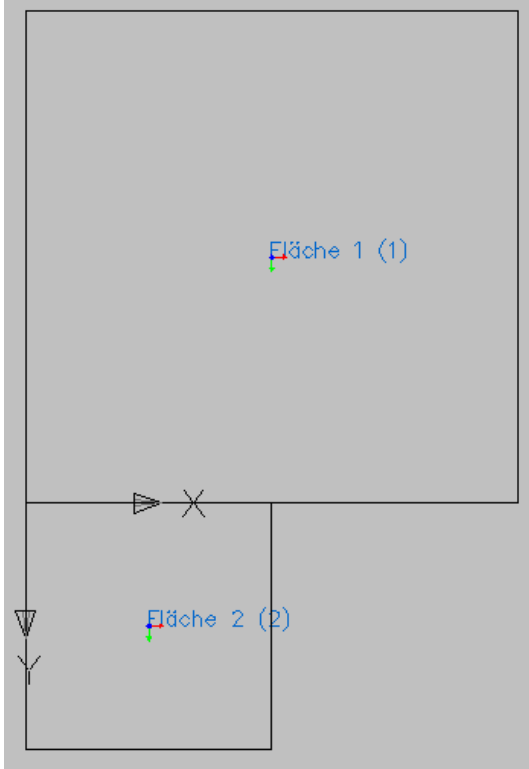
To create a beam 1/3 of the length of an edge, subdivide the edge two times, remove the support conditions and assign a cross section to the middle section.

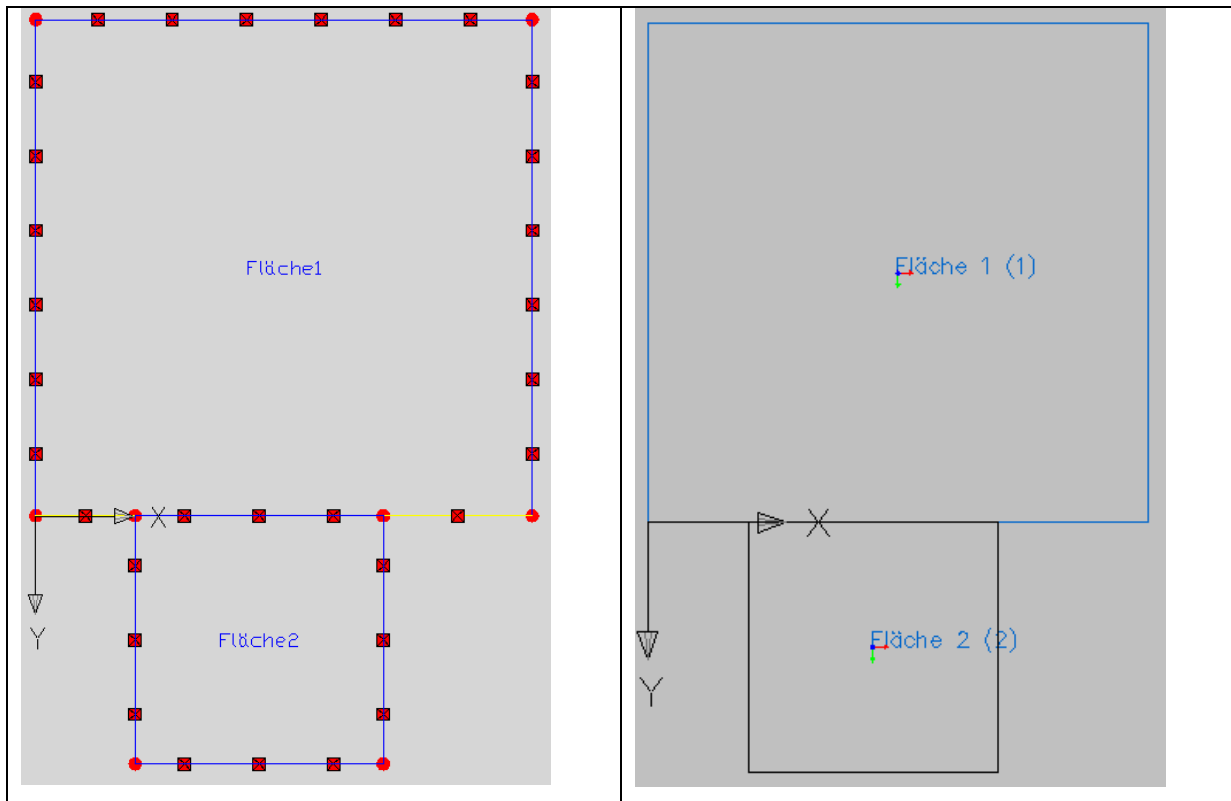


To create a beam 1/3 of the length of an edge, draw a structural line with a cross section at the required location.

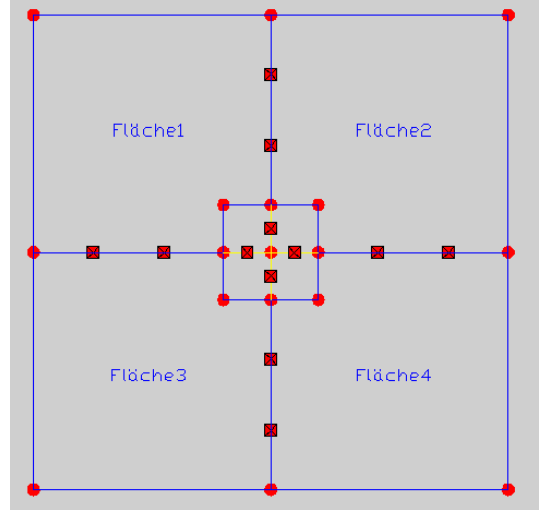
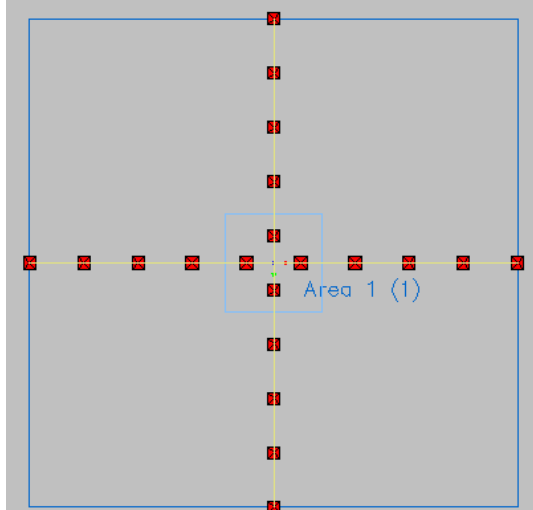


## 1.2 Create a second area, smaller than the first

Previous Approach	New Approach
Results in 7 points, 8 lines and 2 areas.	Results in 2 areas.
	
The boundary edges of the areas have to be split. In large 3D systems the required automatic checks could fail and result in invalid areas.	Intersection of the areas is done by a new mesh generator (SOFiMSHC).
Moving one of the areas basically required redefining both areas as the split operations would be too complex to resolve interactively.	Both areas can be moved around freely without having to split the boundary edges.

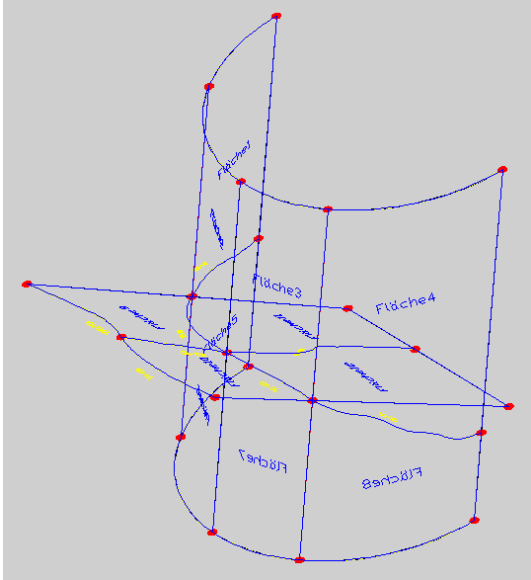
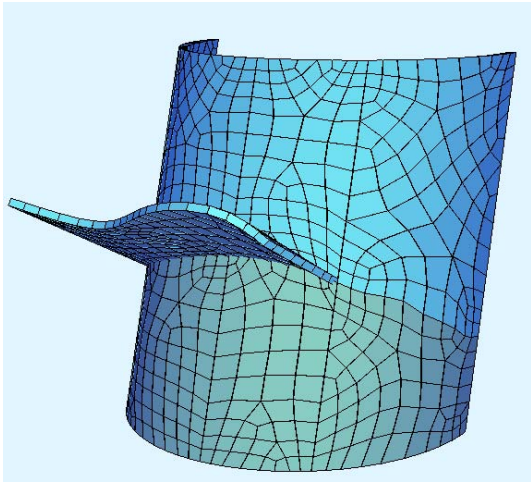
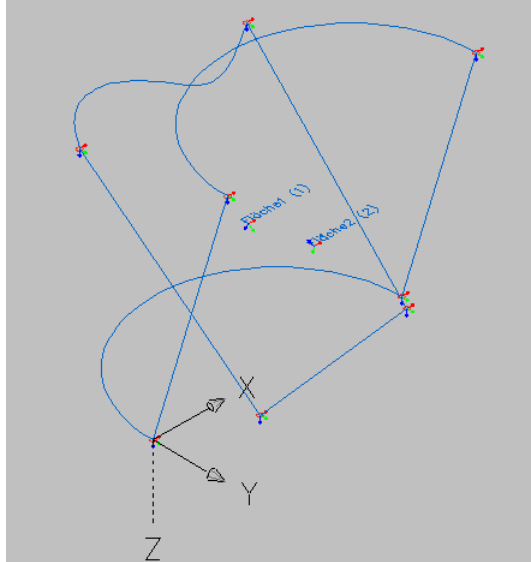
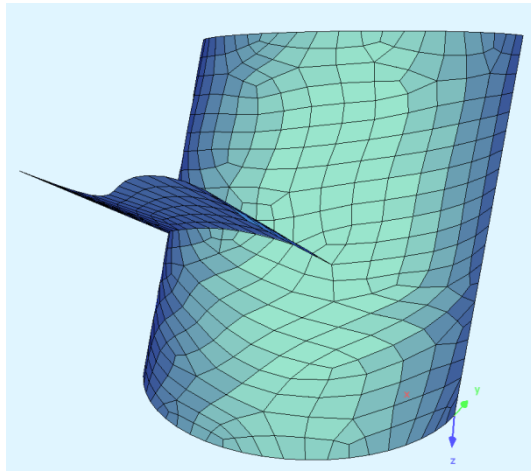


### 1.3 Openings in slabs at the intersection of 4 walls

Previous Approach	New Approach
<p>The opening had to be modeled manually. The support conditions had to be modified for the lines resulting from the subdivided boundary edges</p> <p>.</p>	<p>Results in 1 area, 1 opening and 4 lines with support conditions.</p> <p>Intersections are generated automatically by the mesh generator.</p>
	

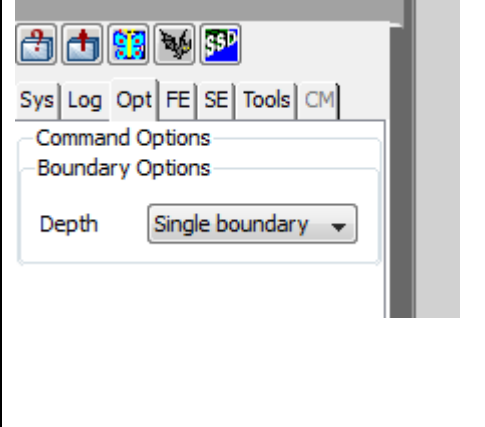
The structural elements now display a number in brackets alongside their name. This is the internal number of the element which is used for defining loading on elements and as a reference in error messages (for example, during mesh generation).

#### 1.4 Intersection of two curved areas

Previous Approach	New Approach
<p>The intersection curves have to be defined manually and then converted into structural lines. Now 12 areas need to be defined by picking the boundary edges.</p>	<p>Create 2 areas by picking 4 curves each. The intersection gets automatically calculated by the mesh generator.</p>
 	 

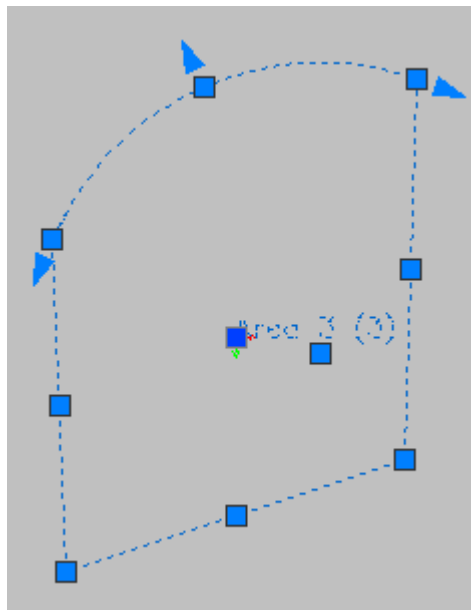
## 1.5 New options for “Point in Area”

When a command with the option “Point in Area” is active then the Sidebar provides the option to activate the detection of “island areas”.

	<p>The following options are available:</p> <p><b>Single boundary</b> Creates the outer boundary. (Default)</p> <p><b>Detect openings</b> Creates the outer boundary and finds all areas within the outer boundary (“islands”) and converts them to openings.</p>
---	---

## 1.6 New grip points for structural elements

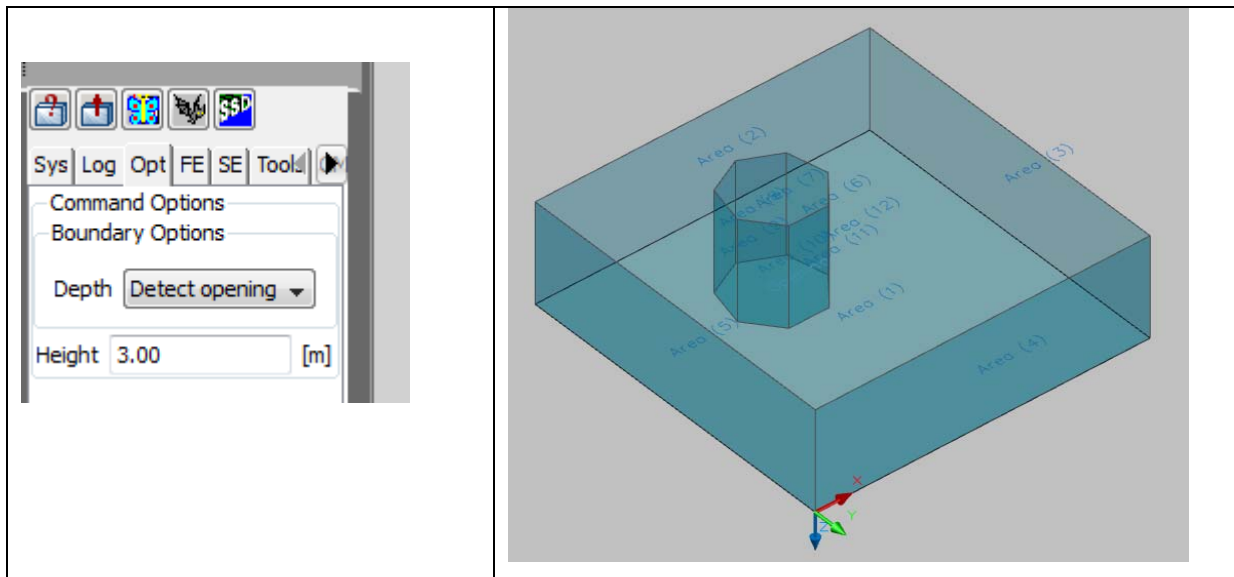
When created from AutoCAD elements the new structural elements inherit the grip points of the original geometry. For example, a boundary edge created from an arc will inherit the arc grip points so that it is possible to change its radius later on.



## 1.7 New commands for structural area elements

### 1.7.1 Wall

Creates structural areas by extruding lines or picked curves (including arcs and splines). The height can be specified in the Sidebar. When using the option “Point in Area” the option to detect openings (“islands”) is available, too.



The command is a helper for generating vertical (or extruded) areas. The generated area elements do not keep any wall specific information (for example, height).

### 1.7.2 Attribute Area

Attribute areas can be used to locally modify a set of properties of a structural area. This feature is not available in the initial release (2010-1).

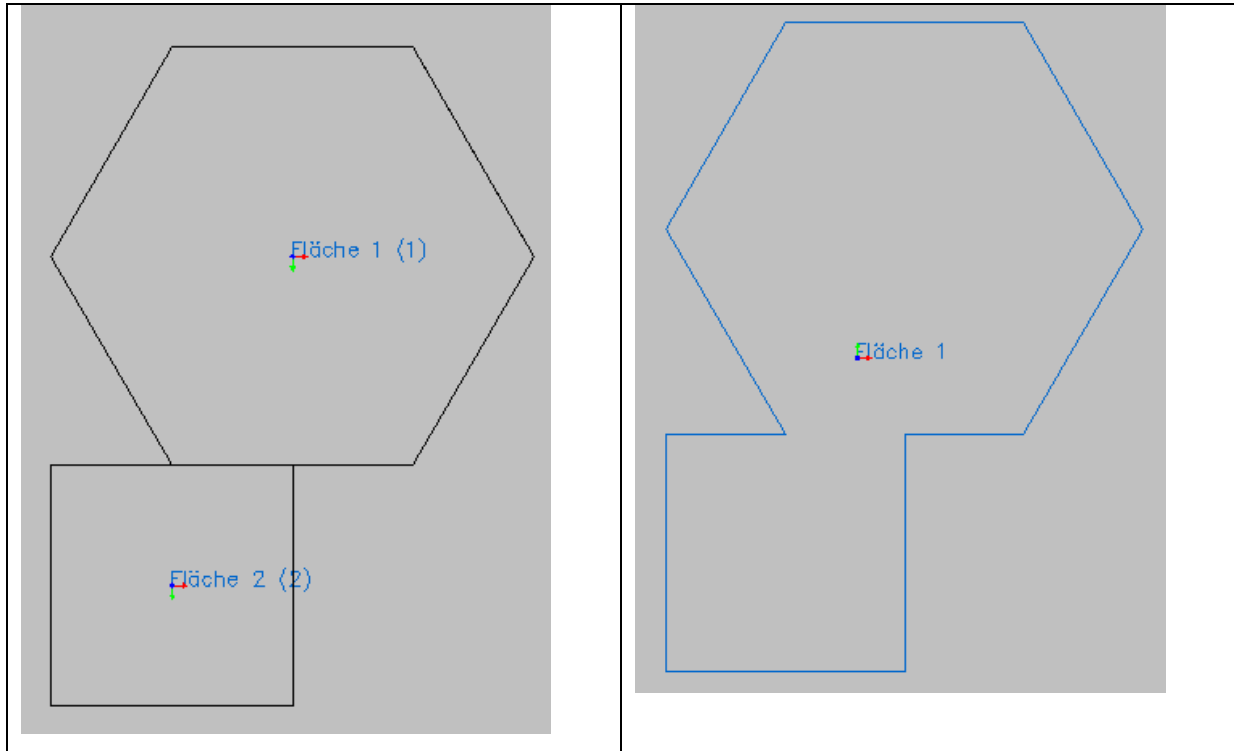
### 1.7.3 Edit structural area edge properties

This command allows to edit the properties of structural area edges (not structural lines!) of one or more structural areas simultaneously.

### 1.7.4 Add areas

This command executes a Boolean operation to combine two areas. Two areas have to be picked. The properties of the first area will be kept, and the geometry of the second one will be added to the first one.

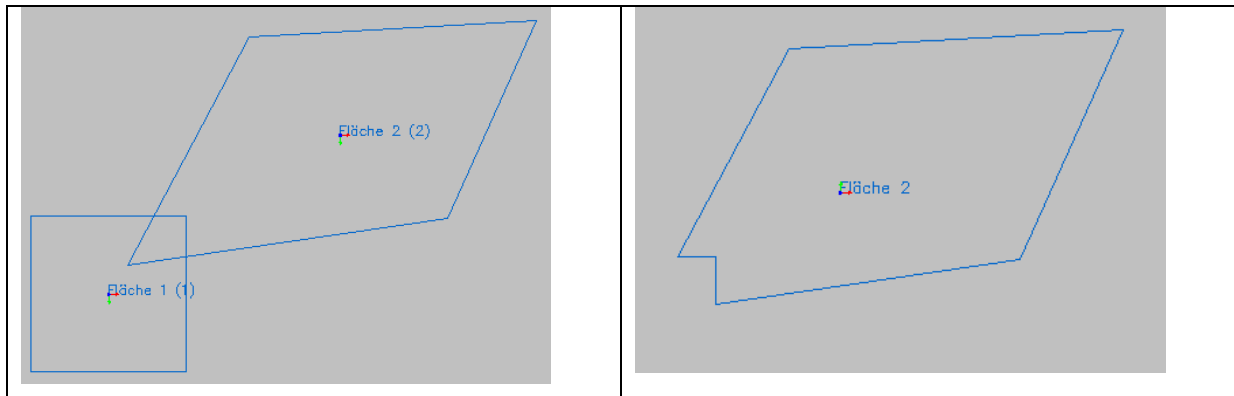
This command works with planar structural areas only.



### 1.7.5 Subtract area

This command executes a Boolean operation to remove the intersection area of two areas from the area first picked.

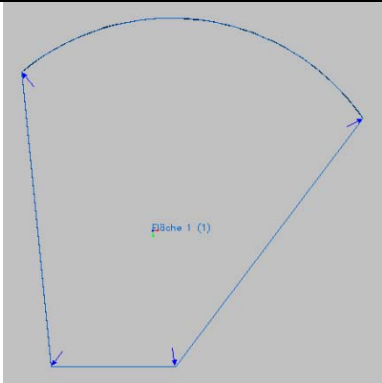
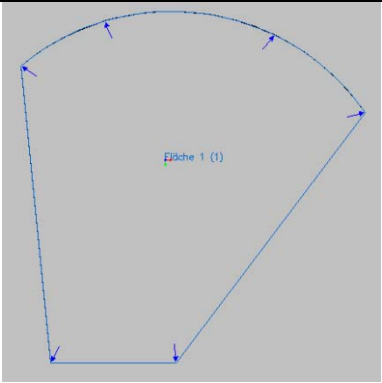
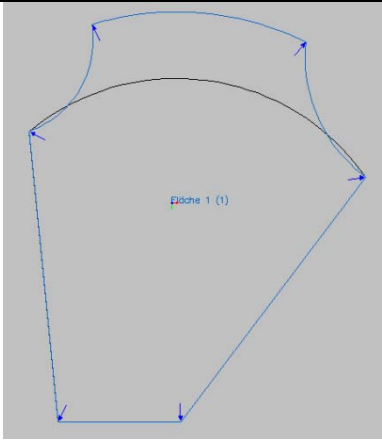
This command works with planar structural areas only.



### 1.7.6 Add point on area boundary edge

In order to modify the geometry of a structural area, it is possible to add additional points to the area boundary. Each additional point subdivides a boundary curve and results in an additional grip point which can be used to modify the geometry.



Original Area	Added two points	Modified area
		

### 1.7.7 Remove point from area boundary edge

Superfluous points can be removed from the boundary of a structural area thereby simplifying the geometry.

## 1.8 New options available for linear or curved structural elements

The dialog box “Modify Structural Line” provides the following meshing options on the tab “Beam/Cable”:

### ***Mesh Automatically***

The structural line will be meshed resulting in several elements. The elements will be connected to an element mesh if one has been defined.

### ***Create Elements***

The structural line will be meshed resulting in several elements which will **not** be connected to an existing element mesh.

### ***Create One Element without Sections***

The structural line will result in a single element which will not be connected to an existing element mesh. This option is useful to define cables.

### ***Create One Element with Sections***

The structural line will result in a single element which will not be connected to an existing element mesh. Beam sections will be generated, though.

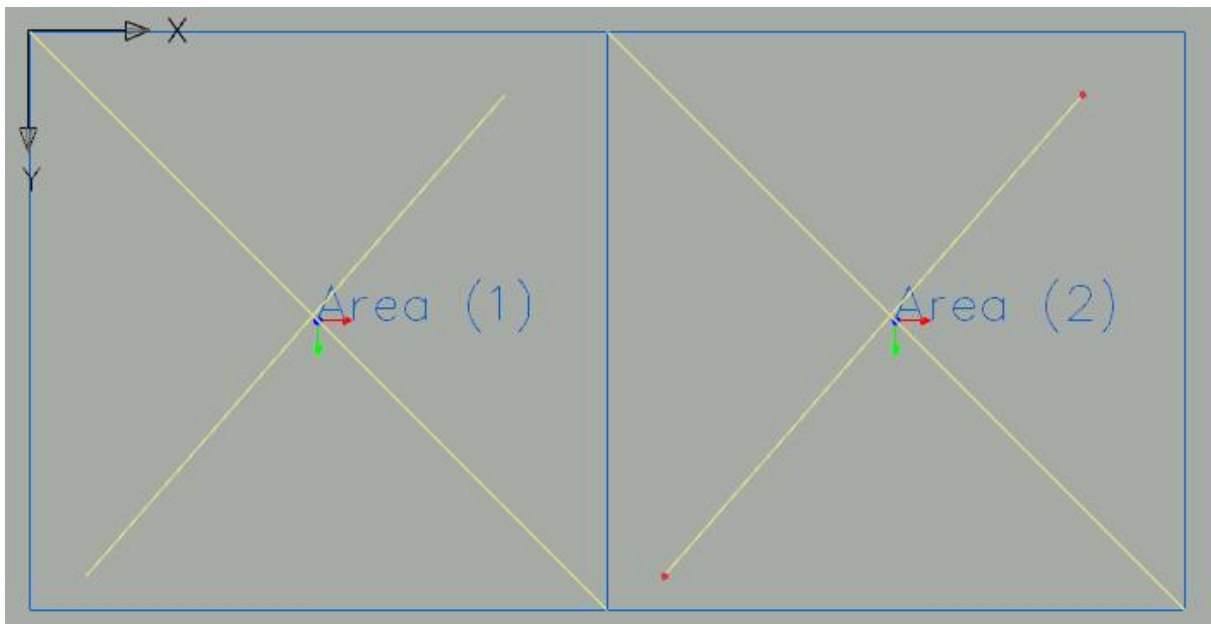
In addition the following option can be used to control meshing behavior:

### ***Intersect With Other Elements***

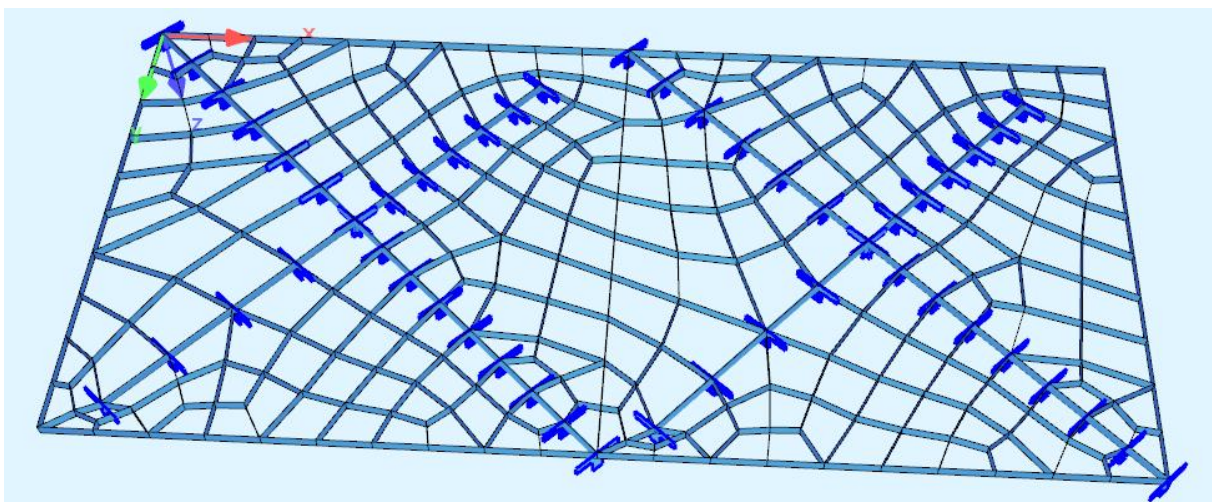
This option avoids intersections between resulting elements and other elements. The elements end points will only be connected to an element mesh if they lie directly on

a structural point, structural line or structural area edge. To connect the end point of a line which is located within a slab to the mesh of the slab it is necessary to create a structural point at the end points of the structural line.

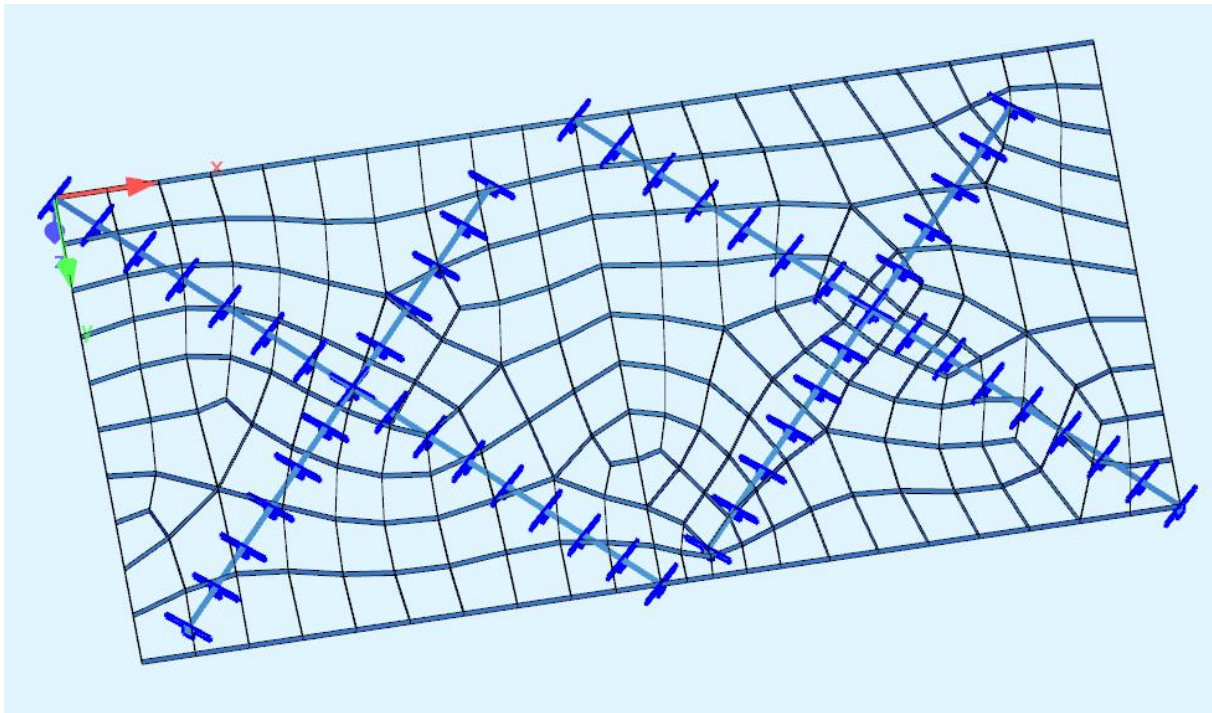
The following pictures are provided to illustrate the behavior. In each case we mesh the same structure except for modifications to the properties of the structural lines.. The slab to the right has additional structural points defined at the end point of the shorter of the two structural lines in order to force a connection to the surrounding element mesh.



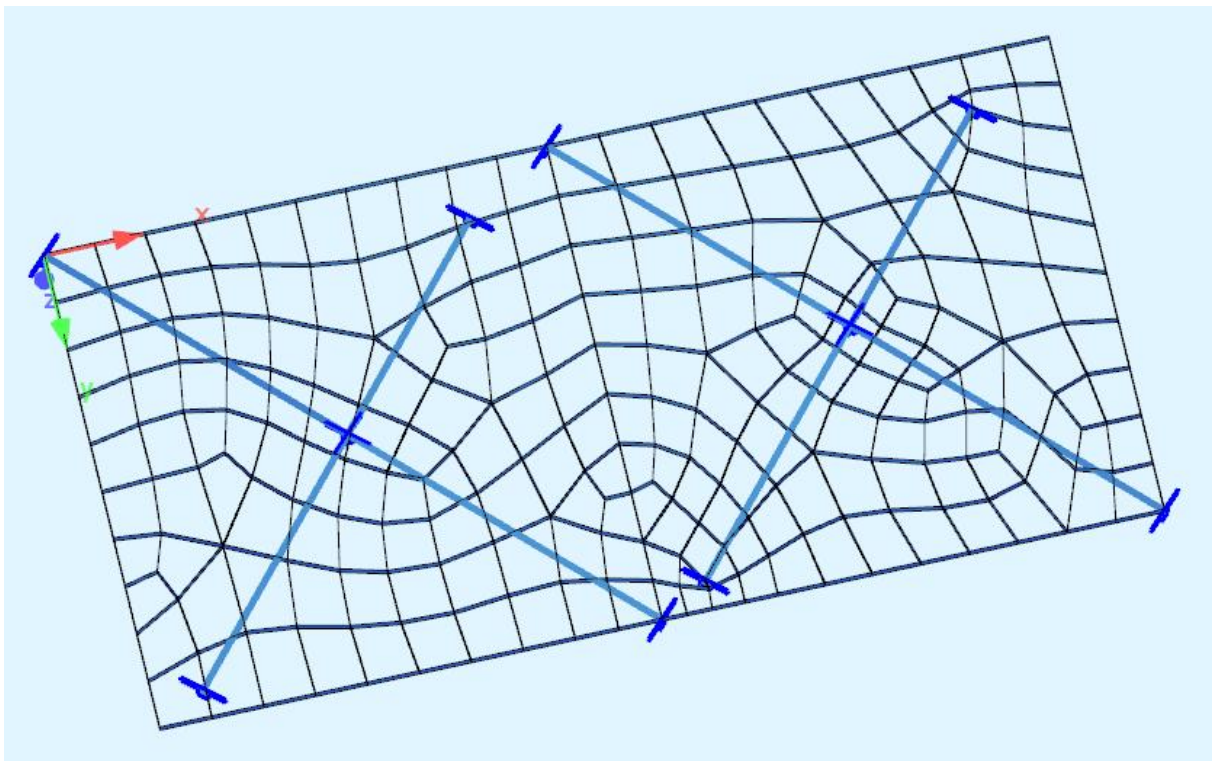
### Mesh Automatically



## Create Elements



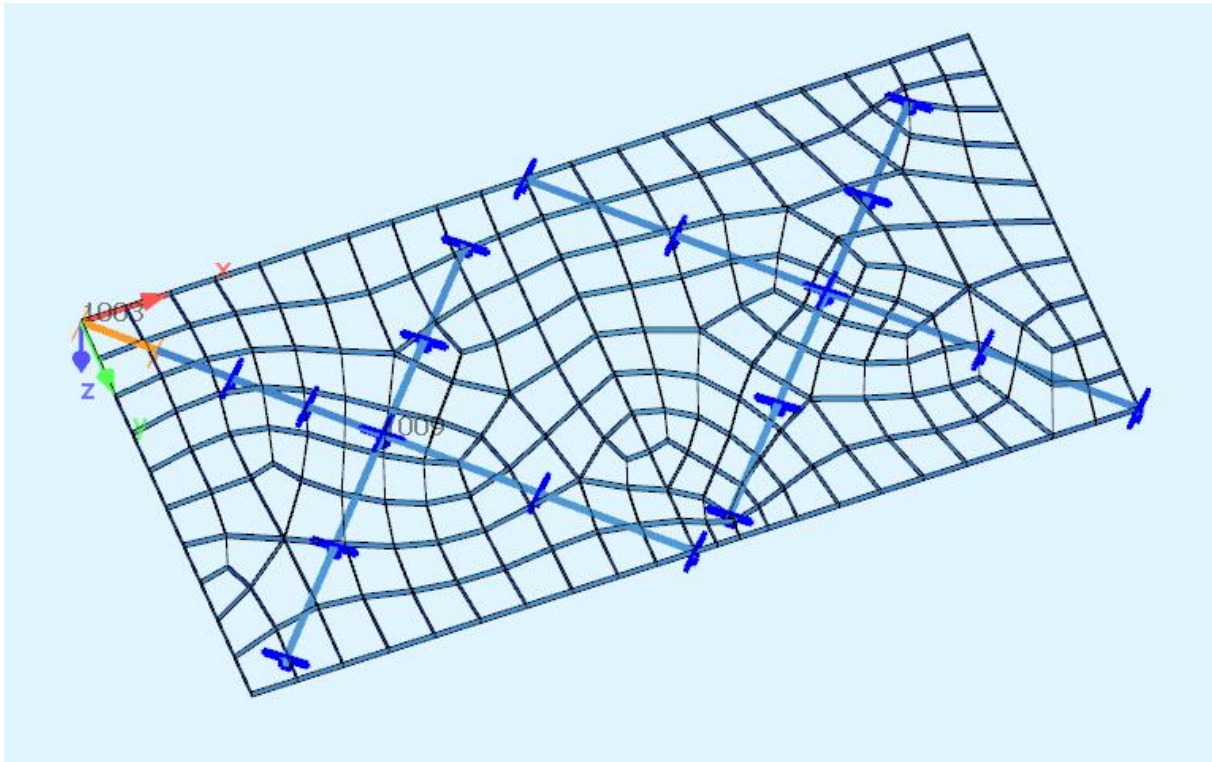
## Create One Element without Sections





## Create One Element with Sections

The node numbers at start and end of a beam have been enabled for this screenshot to more clearly illustrate the behavior.



## 1.9 Tables in element dialogs

### 1.9.1 General

Starting with the new FEA programs (version 2010) it is now possible to define constraints and springs in more than one direction. To improve editing workflow the element dialogs of structural points and lines provide new tabs with tables.

The toolbar to the right provides tools for adding, removing and copying entries in the table. Copying when one or more entry is selected results in copies of all the selected entries.

### 1.9.2 Constraints

The tab “Constraints” provides in its table a column “Element” which allows to pick another structural element in the AutoCAD model view to define a target for the constraint to be defined. If the target structural point does not yet exist a structural point with default properties will be created automatically.

## 2 Loads

### 2.1 New dialog tab in dialogs of structural elements

The dialogs of all structural elements have a new tab “Loads” which provides a table to edit loads specific to the selected element(s).

### 2.2 Multiple load cases per free load

Free loads can be assigned to several load cases at once by specifying a comma-separated list of load cases numbers.

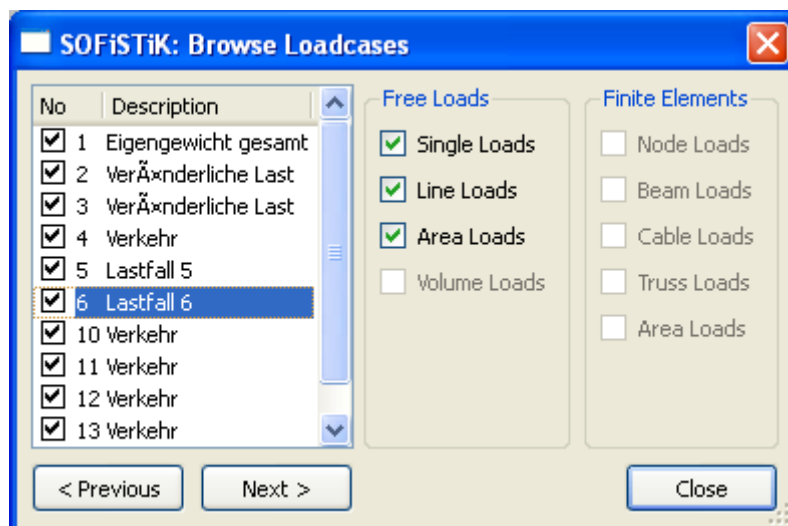
### 2.3 Multiple reference elements

Free loads can reference several groups or elements at once by specifying a comma-separated list of group numbers or element numbers respectively.

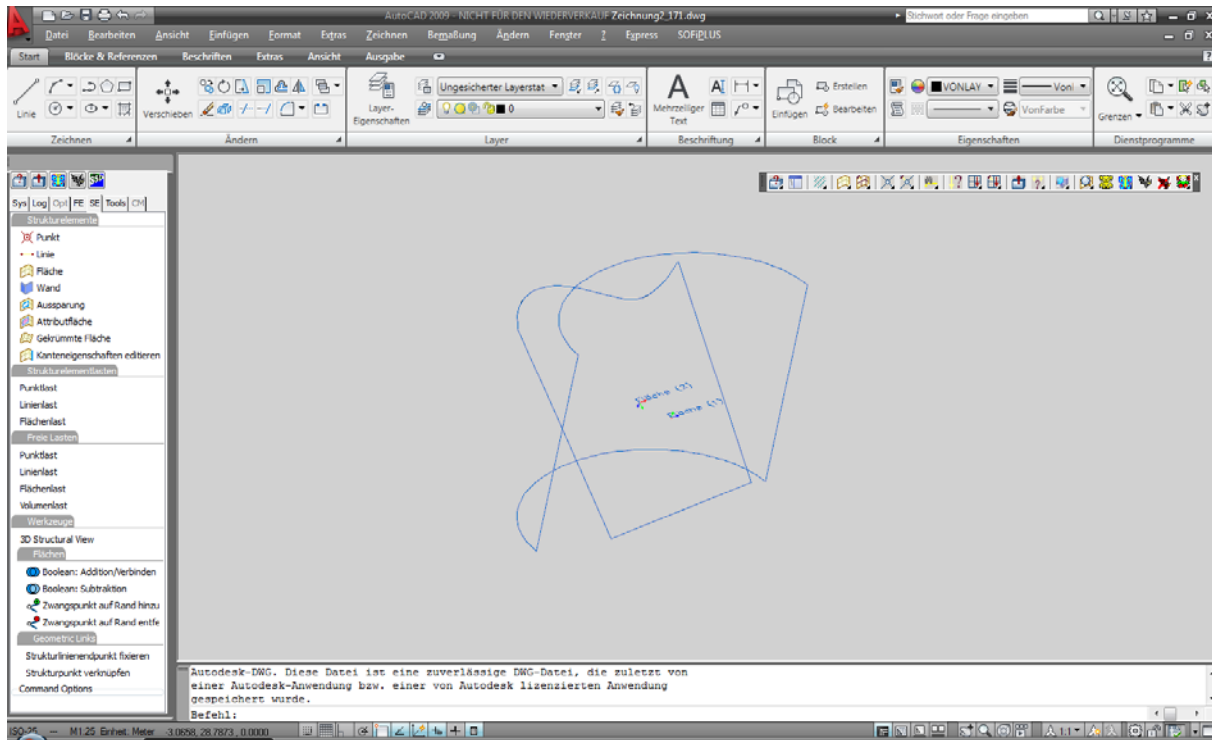
### 2.4 Load Case Browser

The visibility of loads within a drawing can be controlled using the Load Case Browser. The load case browser is a non-modal dialog whose changes are applied to the drawing immediately.

When opening the load case browser the settings in the dialog represent the currently active visibility settings for loads in the current drawing.



### 3 Sidebar



The sidebar provides several tabs including a project explorer (materials, cross sections and more), a log view (for warnings and errors during mesh generation) and several tabs with SOFiPLUS commands.

A right-click on an entry in the log view brings up a list of affected elements (if available) and a click on an entry in this list brings the element into view in the currently active viewport.

### 4 Named Selections

The commands “Display All” and “Display Selection Only” could be used to control the visibility of elements in a drawing.

As of version 2010 currently active visibility settings can be saved as named selections. The commands for saving and restoring named selections are available in the general right-click menu (option “Named Selections”).

When restoring a selection elements that have been created after the selection has last been saved will be displayed.

## **5 Cross Section Editor**

## **6 New Commands**

### **6.1 3D Structural View / 2D Wireframe View**

The command “3D Structural View” enables a 3D shaded environment (with configurable translucency) designed to ease working with large 3D structures.

The command “2D Wireframe View” returns the working environment to the default AutoCAD drawing environment.

### **6.2 Strukturlinienendpunkt mit Strukturpunkt verknüpfen**

### **6.3 Strukturpunkt verknüpfen**

## **7 Zusammengelegte Befehle**

Es gab eine Reihe von Befehlen, die im Grunde dasselbe erledigten, aber für jedes Element einzeln zur Verfügung stand. Diese Befehle wurden nun zusammengelegt.

Dies sind (bzw. sollen werden):

- Elemente suchen
- Elemente ausrichten
- Elemente umnummerieren