

Contain

Cross-Section.....	1
Primitives.....	2
General information.....	2
Host primitive.....	2
Prestressing.....	3
Templates.....	4
Actions.....	5
Materials	6
Elastic	6
Concrete	8
Reinforcement.....	9
Carbonfiber	11
Partial safety factors	12
Behavior diagrams.....	13
Moment-curvature diagram.....	13
Interaction diagram.....	14

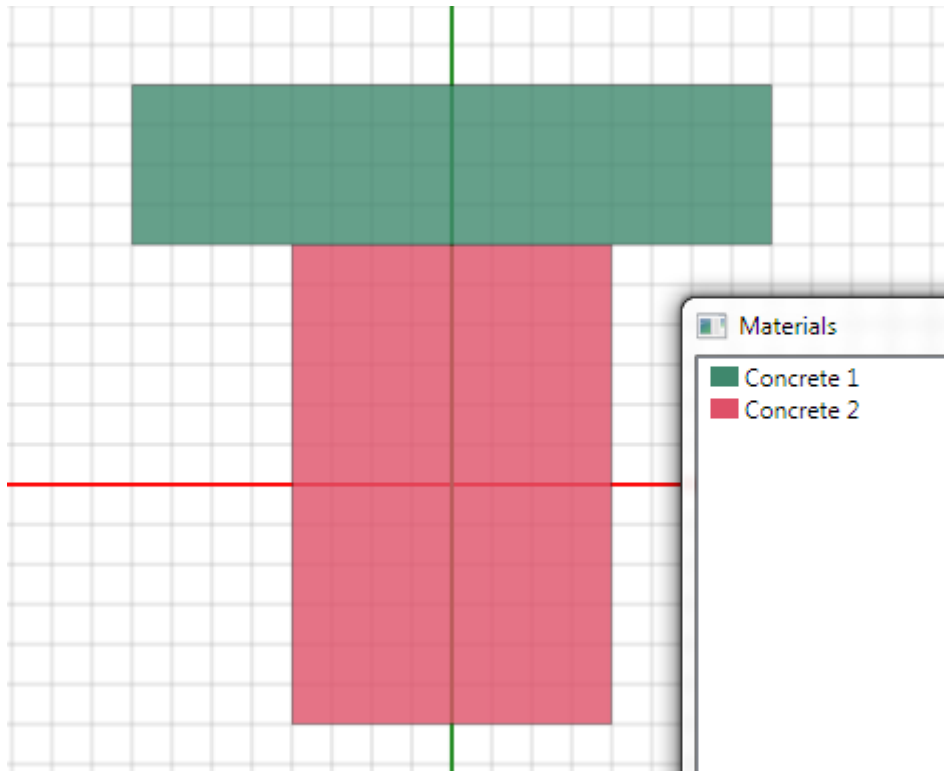
Cross-Section

Cross-section of beam, according to assumptions of strength of material, characterize geometry of infinitely small part of beam,

As usual, cross-section in Structure Helper includes next groups:

- Actions (loads, for example),
- Primitives,
- Calculators.

Each cross-section may contain any amount of those primitives, for instance, you can create T-shape beam from two different primitives – rectangle of web and rectangle of flange of T-beam. Note, than you can assign different material for each of those primitives and obtain combined structure if you want that.



Composite cross-section of T-shape with 2 different materials

Primitives

General information

Primitives are just simple geometry objects. Currently, you can use few types of primitives, as follow:

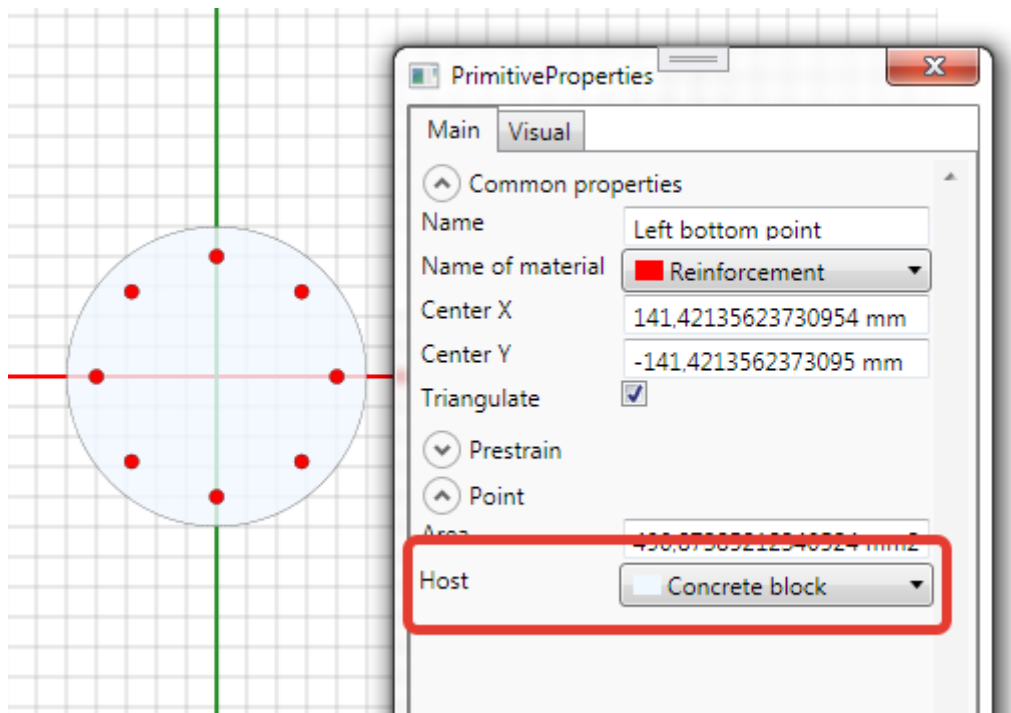
- Rectangle,
- Circle,
- Point,
- Reinforcement point.

There are number of ways to create new primitives, for instance you can create new primitive via main menu, context menu of Geometry panel or via context menu of work plain of cross-section.

Therefore, you can use templates for adding number of standard primitives with their materials.

Host primitive

Some primitives (rectangle and circle) may play a role a host for other ones. Term “host” means, that main primitive contain any count of slave primitives. For example, when you create cross-section of reinforced concrete column, you rebars will have a host – rectangle or circle with concrete material. Host primitive is required to gain many important properties of concrete when RC member is calculated, for instance for obtaining of lapping length for rebars.



Host primitive for rebar

Prestressing

Prestress is a widely known way to create initial (before applying a load) stress or strain in structures. Since Structure Helper operate with not only linear elastic material, it is assumed, that you have to assign initial strain instead initial stresses. In the simplest case of linear elastic material you can calculate initial strain (prestrain) from follow expression:

$$\varepsilon_{initial} = \frac{\sigma_{initial}}{E}$$

In Structure Helper you can assign 3 components of initial strains (prestrain):

- Curvature about X-axis (Kx),
- Curvature about Y-axis (Ky),
- Strain along Z-axis.

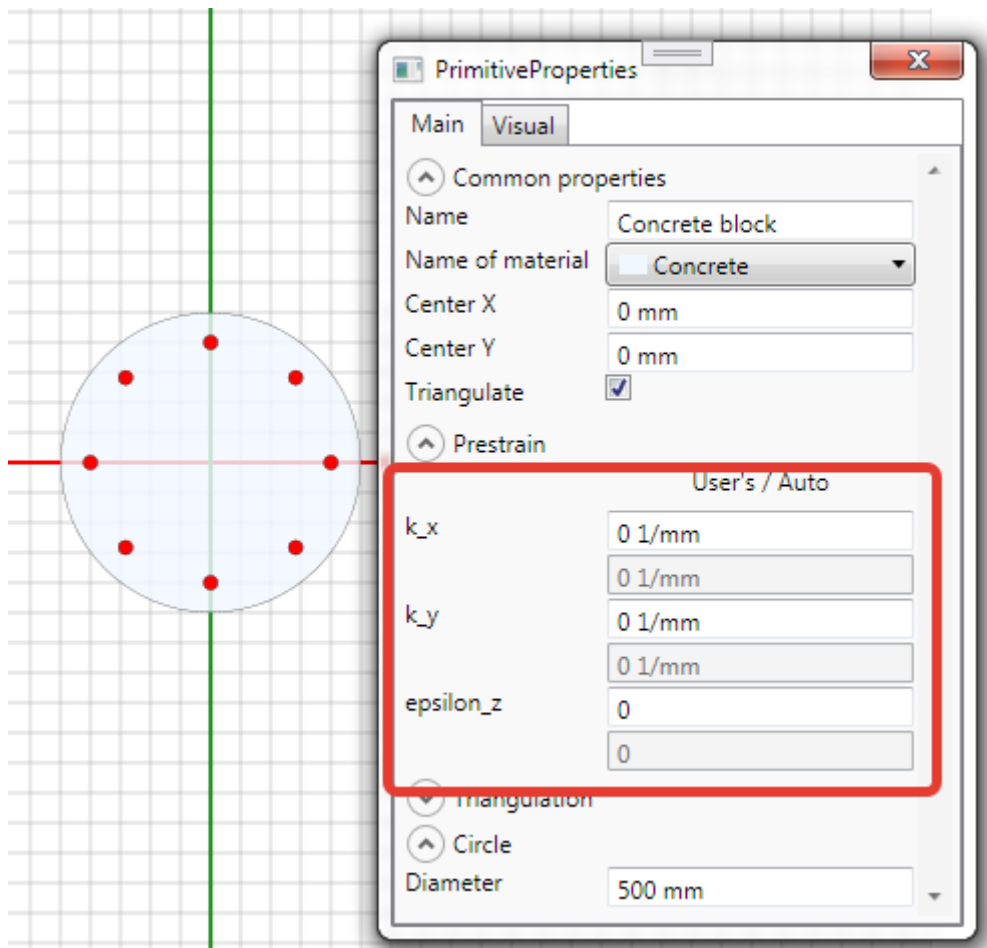


Fig. Prestrain for primitive

Prestrain may be assigned for any type of primitive and any material, for instance, you can consider for concrete and reinforcement simultaneously, also you can consider different value of prestrain for each of rebars.

After triangulation of primitive, for every elementary part of fiber model will be assigned prestrain as below:

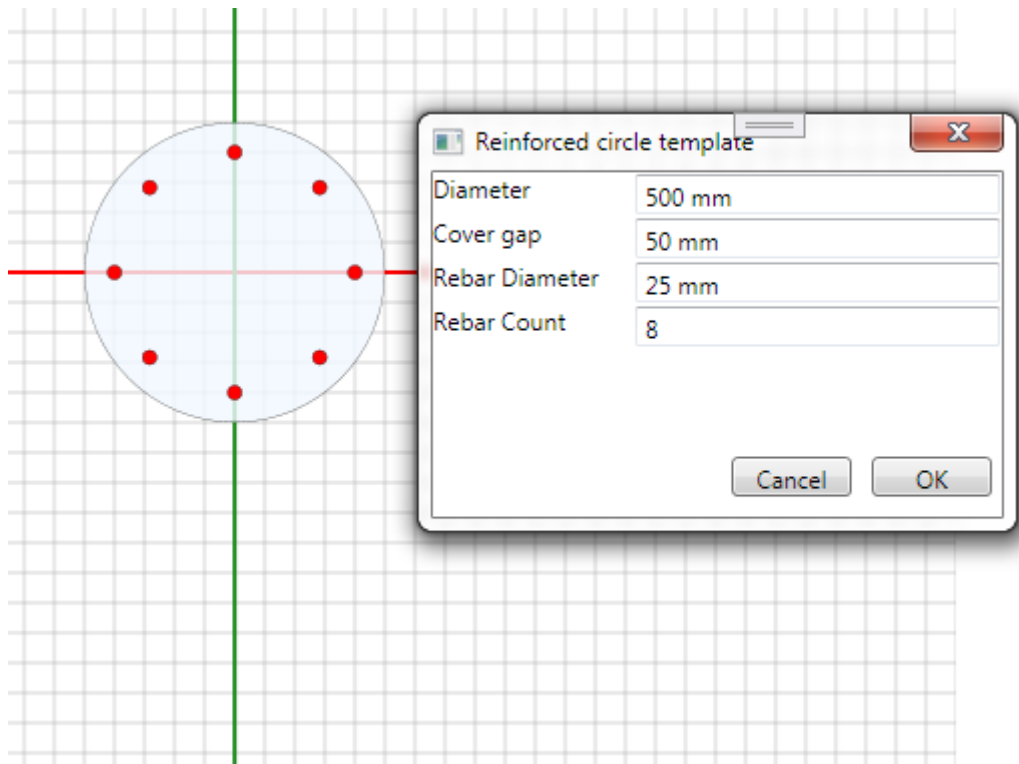
$$\varepsilon = K_x \cdot y + K_y \cdot x + \varepsilon_z,$$

where x and y – coordinates of elementary part along x-axis and y-axis respectively.

Templates

Sometimes, it is quite difficult to create primitives manually with their coordinates and geometry properties like width and height. Templates are extremely useful tools for simplification of creating of primitives, materials, actions and calculators. For instance you can use template of RC circle-shape element instead of manual calculation of coordinates of each rebar in cross-section.

Also, template creates example of action, materials and calculator. As result, you can solve your problem instantaneously after using template. Sure, using of template doesn't restrict your opportunities for manual creating of primitives, materials etc., and you can use number of templates one by one if you want.



Template of circle-shape or RC member

Currently, you can use next kinds of templates:

- Template of rectangle RC beam,
- Template of rectangle RC column,
- Template of rectangle RC slab (with default width 1m).
- Template of circle RC member.

Actions

You can create many different actions and combine them in calculation as you want.

In actual version of Structure Helper you can use 2 types of action:

- Force action for user defined combination of force,
- Force action by factor of long term load and partial safety factor for ultimate limit state.

In fact, calculator uses load set which it obtains from some actions. As usual, load set includes 4 different combinations – one for short term load, second one for long term load and both of them for service limit state (SLS) and ultimate limit state (ULS).

User defined force action is used when you need to consider many different combination which will combined in one load set. Thus, this type of action, as rule, requires to assign 12 values for each action (3 components of force – M_x , M_y , N_z for each of 4 combinations).

The second type of action requires significantly less number of values – 3 value of force and 2 factors (one for long term load and second one as partial safety factor for ULS). On the other hand, this kind of action is suitable only for situations, where all 4 combination of result load set will be proportional one to another.

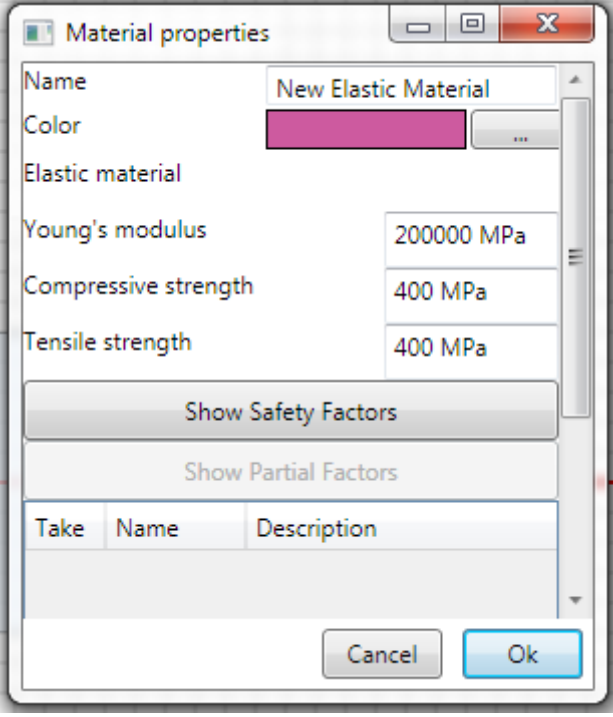
Materials

Material in Structure Helper is a way to describe mechanical behavior of real materials in mathematical expressions. Thus, main property of material in Structure Helper is a diagram of that material, which describes function of stresses depending on strains. Structure Helper's calculation methods are based on hypothesis of plain cross-section and do not take into account large deformations.

For your purposes you can use prescribed materials from library (like concrete and reinforcement materials) or create your own linear material with limit stress.

To estimate behavior of material you can see diagram of this material

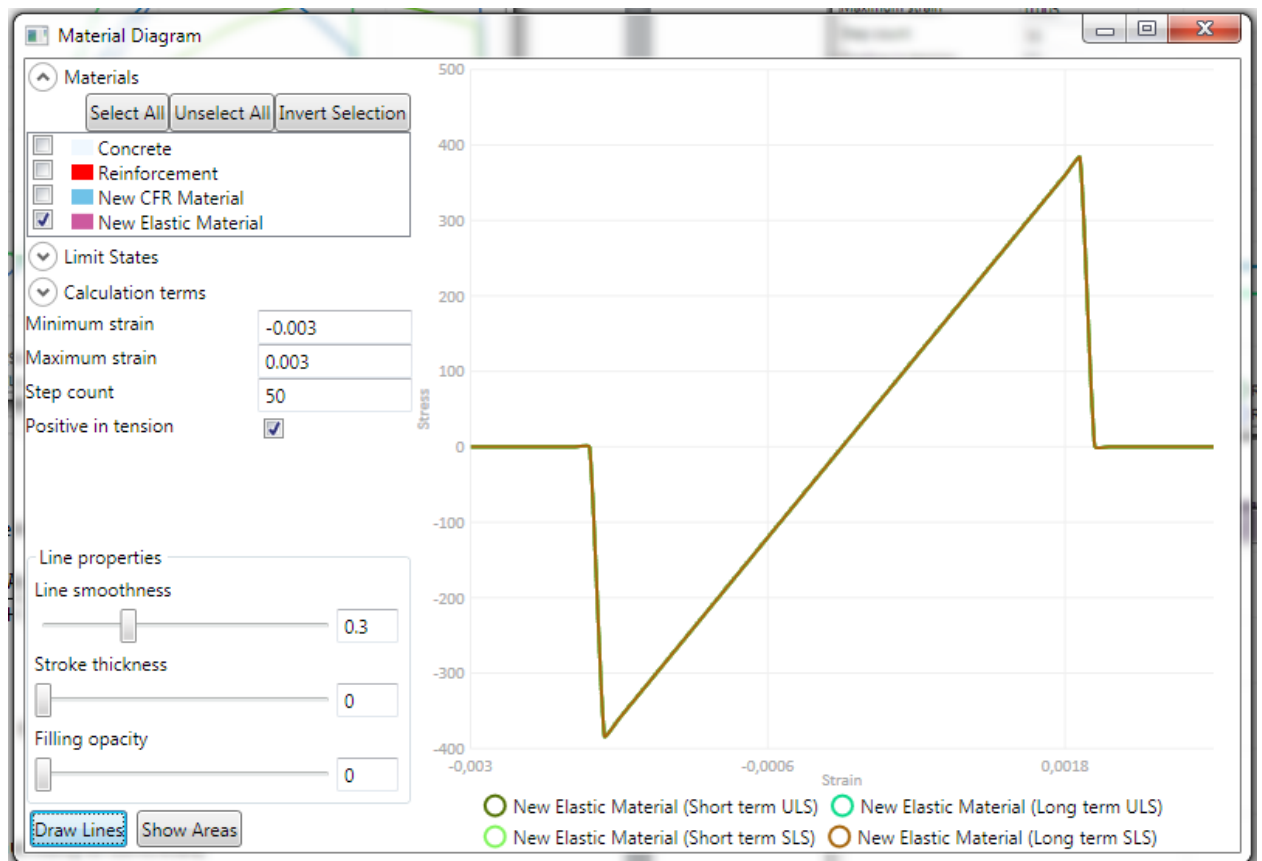
Elastic



The screenshot shows a software window titled "Material properties". It contains several input fields and buttons. The "Name" field is set to "New Elastic Material". The "Color" field shows a pinkish-purple swatch. The "Elastic material" checkbox is checked. The "Young's modulus" is set to "200000 MPa", "Compressive strength" is "400 MPa", and "Tensile strength" is "400 MPa". Below these are two buttons: "Show Safety Factors" and "Show Partial Factors". At the bottom is a table with columns "Take", "Name", and "Description". The table is currently empty. "Cancel" and "Ok" buttons are at the bottom right.

Take	Name	Description
------	------	-------------

Elastic material properties



Stress-strain diagram for elastic material with restriction of stress

Concrete

Material properties

Name

Concrete

Color

...

Material Code

GOST 26633-2015

Material Kind

B40

Material Model

Curve

Tension for ULS

☐

Tension for SLS

☒

Relative humidity

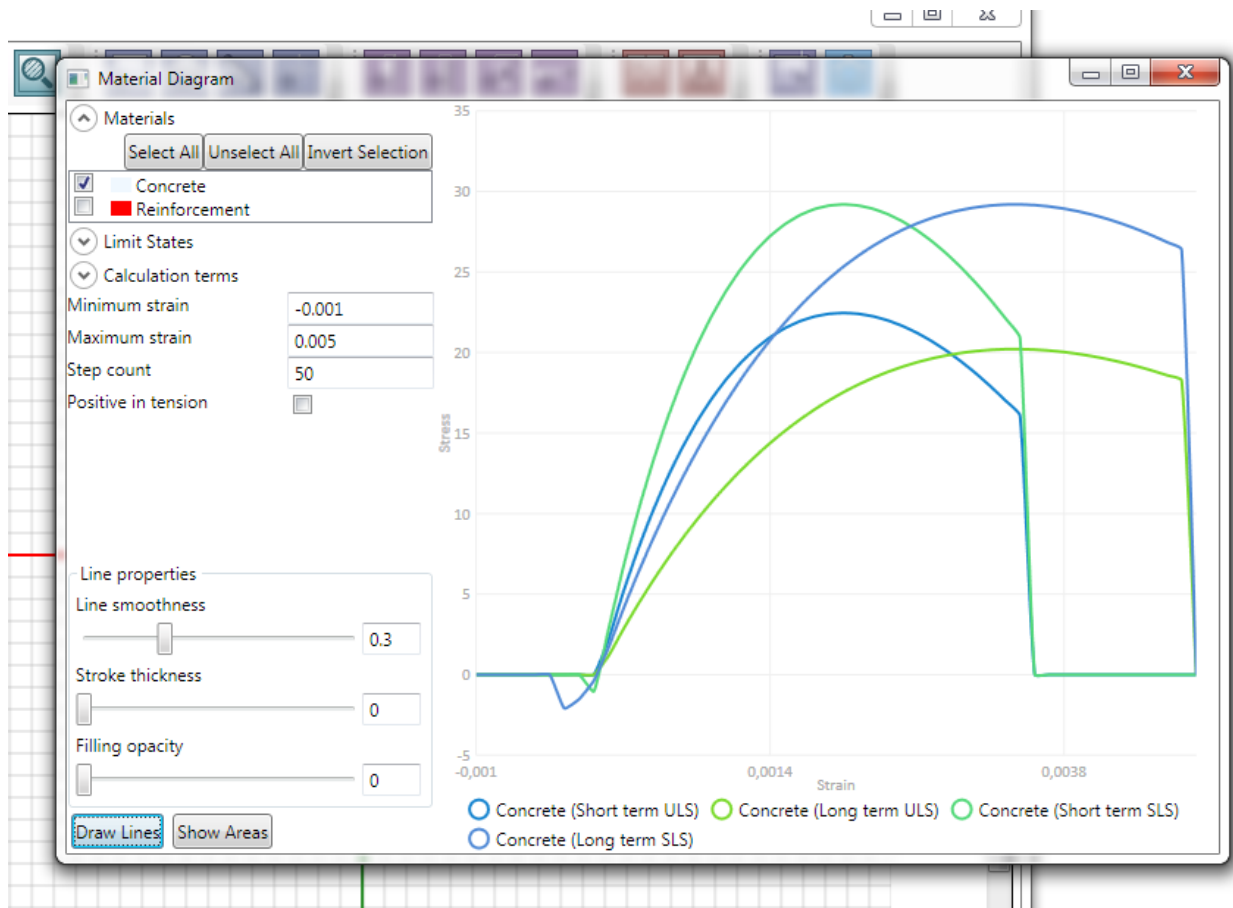
0.55

Show Safety Factors

Take	Name	Description
<input checked="" type="checkbox"/>	Gamma_b1	Coefficient for considering long
<input type="checkbox"/>	Gamma_b2	Coefficient for plain concrete str
<input type="checkbox"/>	Gamma_b3	Coefficient for considering bleed

CancelOk

Concrete properties



Stress-strain diagram for concrete

Nowdays, the only model, available in StructureHelper is curve model obtained from next expression:

$$\sigma = \frac{k\eta - \eta^2}{1 + (k - 2)\eta} \times f_{cm}$$

Where

$$\eta = \frac{\varepsilon}{\varepsilon_{c1}}$$

$\varepsilon_{c1} = 0.002$ –strain corresponding peak stress for short term loading

For long term loading ε_{c1} depends on relative humidity of concrete

$$k = \frac{E_c}{E_{c1}}$$

$$E_{c1} = \frac{f_{cm}}{E_{c1}}$$

Reinforcement

There are 2 available codes for reinforcement material:

GOST 34028-2016 for hot rolled steel, and

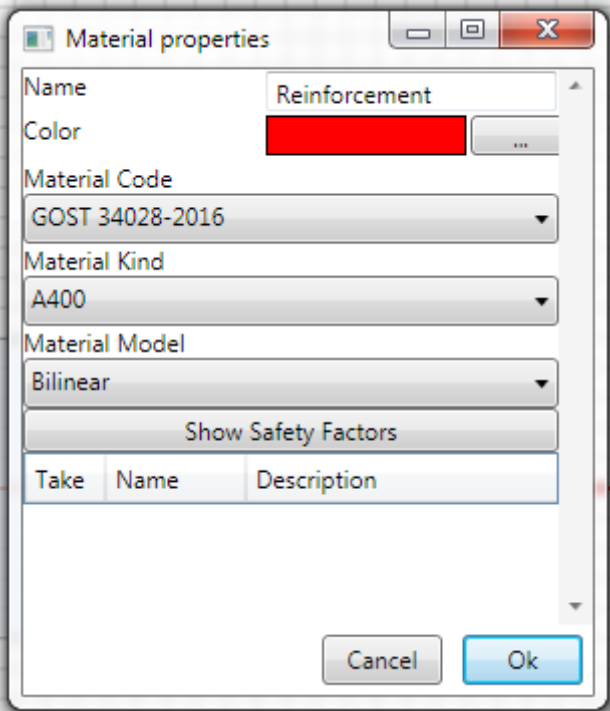
GOST 53772-2010 for high strength cold deformed wires

There are 2 models for reinforcement steel:

Bilinear (with ideal plastic yielding),

Triplelinear

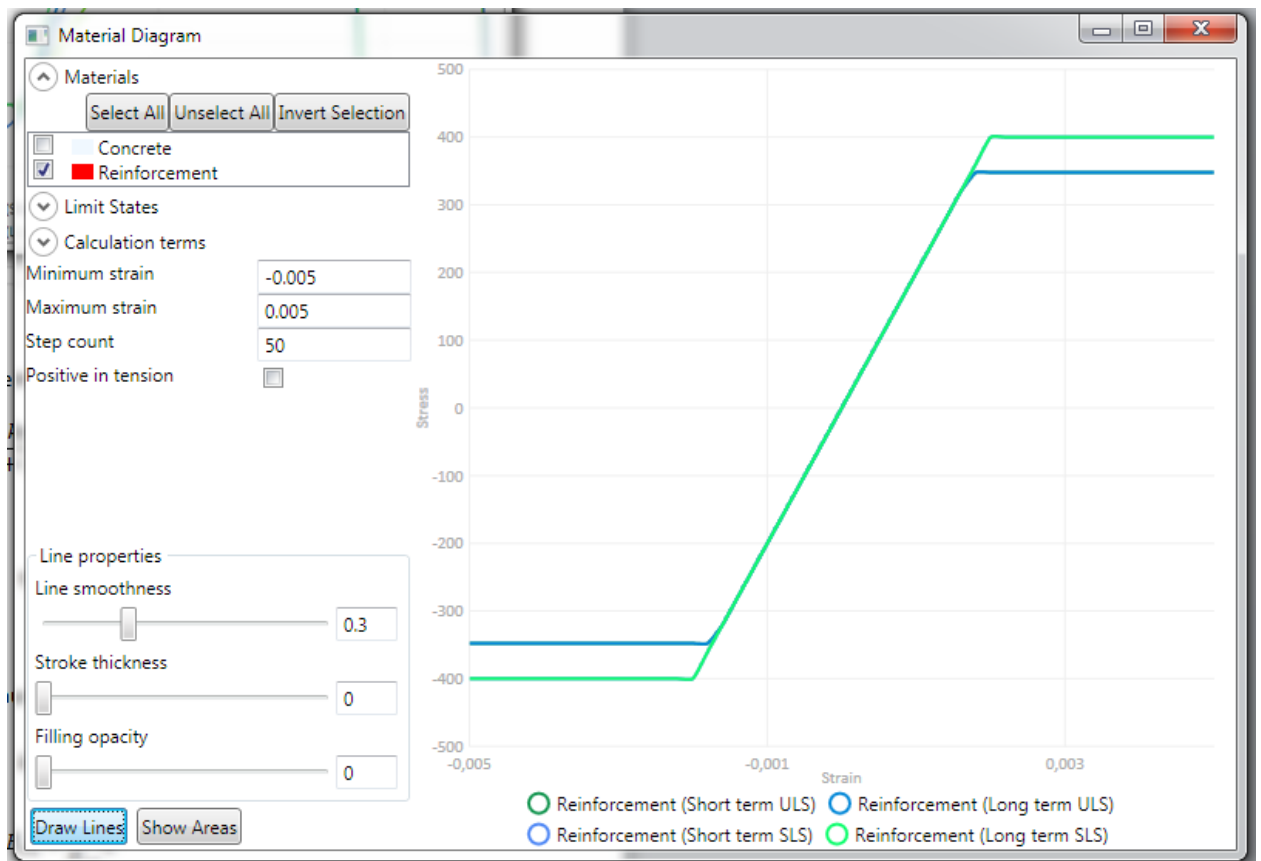
Note, both model are available for any type of reinforcement steel, please choose model type, which is more suitable for your reinforcement type depending on its mechanical properties.



The image shows a software dialog box titled "Material properties". It contains several input fields and dropdown menus for configuring reinforcement steel properties. The "Name" field is set to "Reinforcement". The "Color" field is a red square. The "Material Code" dropdown is set to "GOST 34028-2016". The "Material Kind" dropdown is set to "A400". The "Material Model" dropdown is set to "Bilinear". Below these fields is a button labeled "Show Safety Factors". At the bottom of the dialog is a table with three columns: "Take", "Name", and "Description". The table is currently empty. At the bottom right of the dialog are "Cancel" and "Ok" buttons.

Take	Name	Description
------	------	-------------

Reinforcement material properties



Stress-strain diagram for reinforcement

Carbonfiber

Material properties

Name: New CFR Material

Color:

Elastic material: ☐

Young's modulus: 120000 MPa

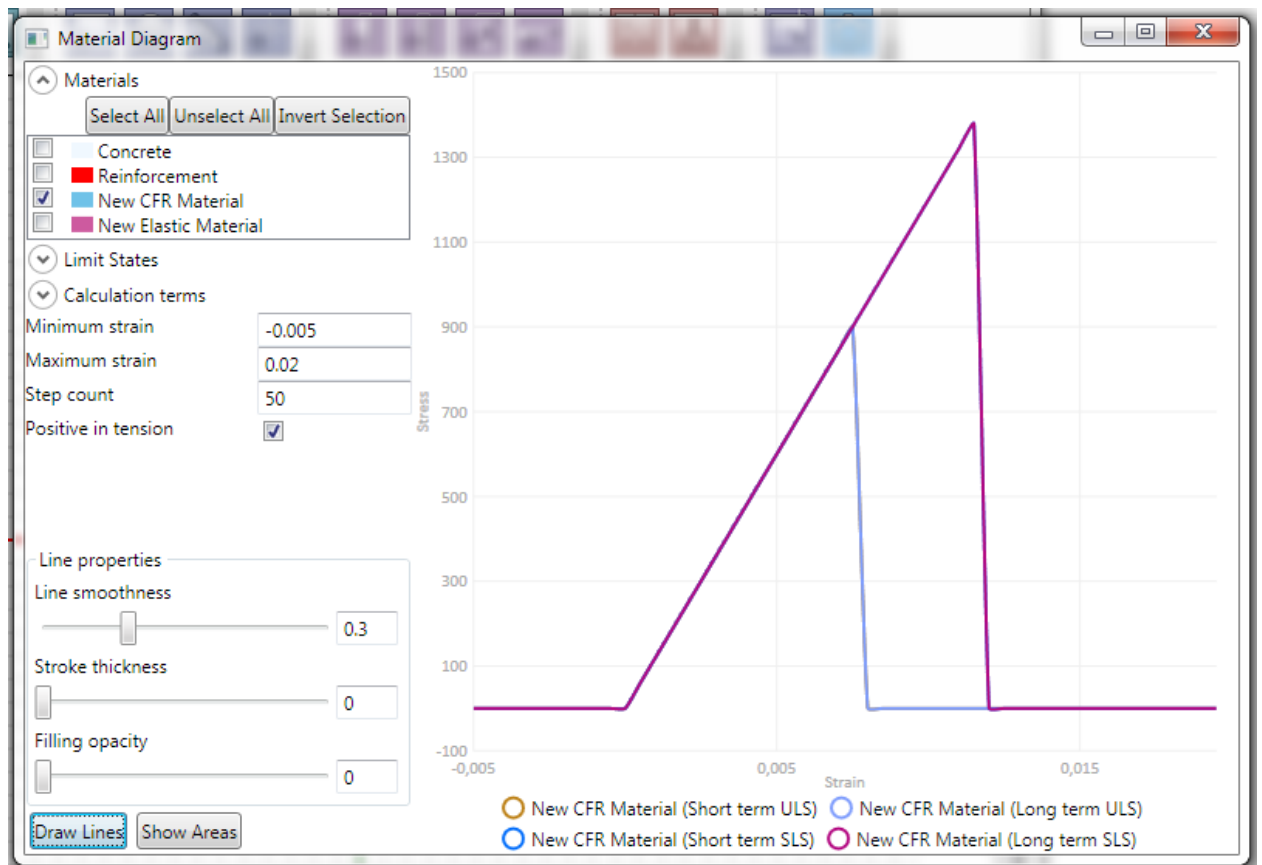
Compressive strength: 0 MPa

Tensile strength: 1400 MPa

☒ Fiber Cohesion Properties

Take	Name	Description
<input checked="" type="checkbox"/>	Gamma_f1	Coefficient for considering environment
<input checked="" type="checkbox"/>	Gamma_f3	Coefficient for considering long term calculation

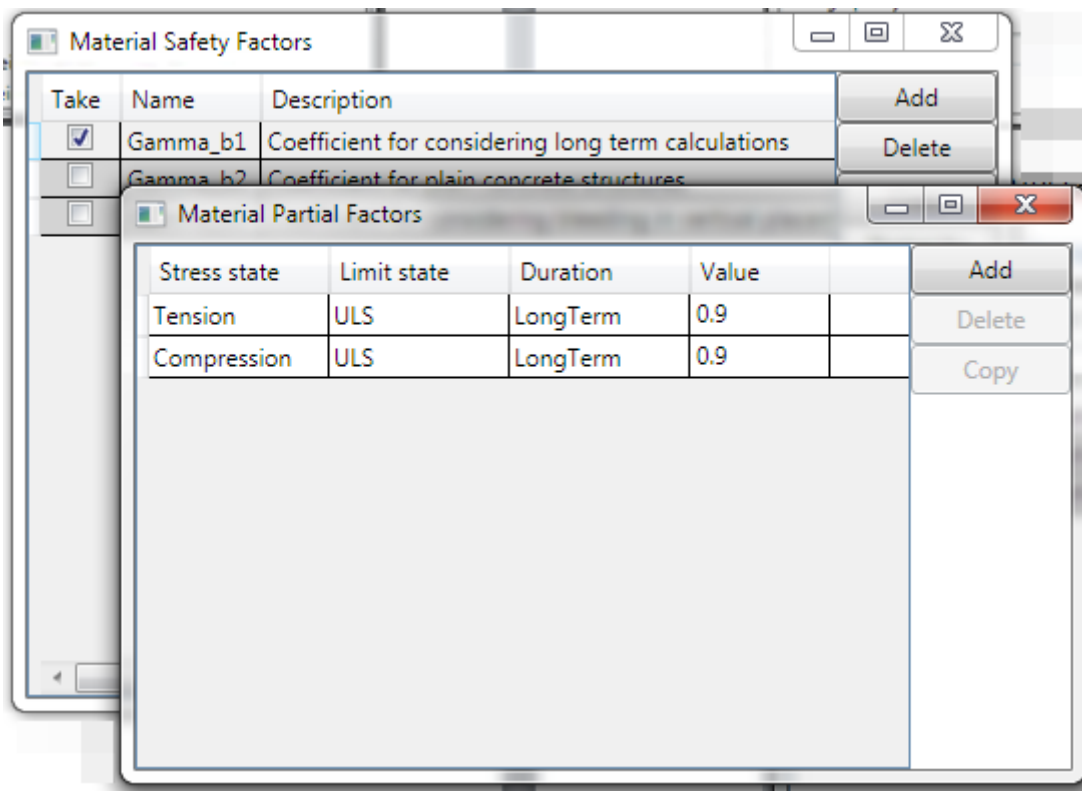
Carbon fiber material properties



Stress-strain diagram for carbonfiber material

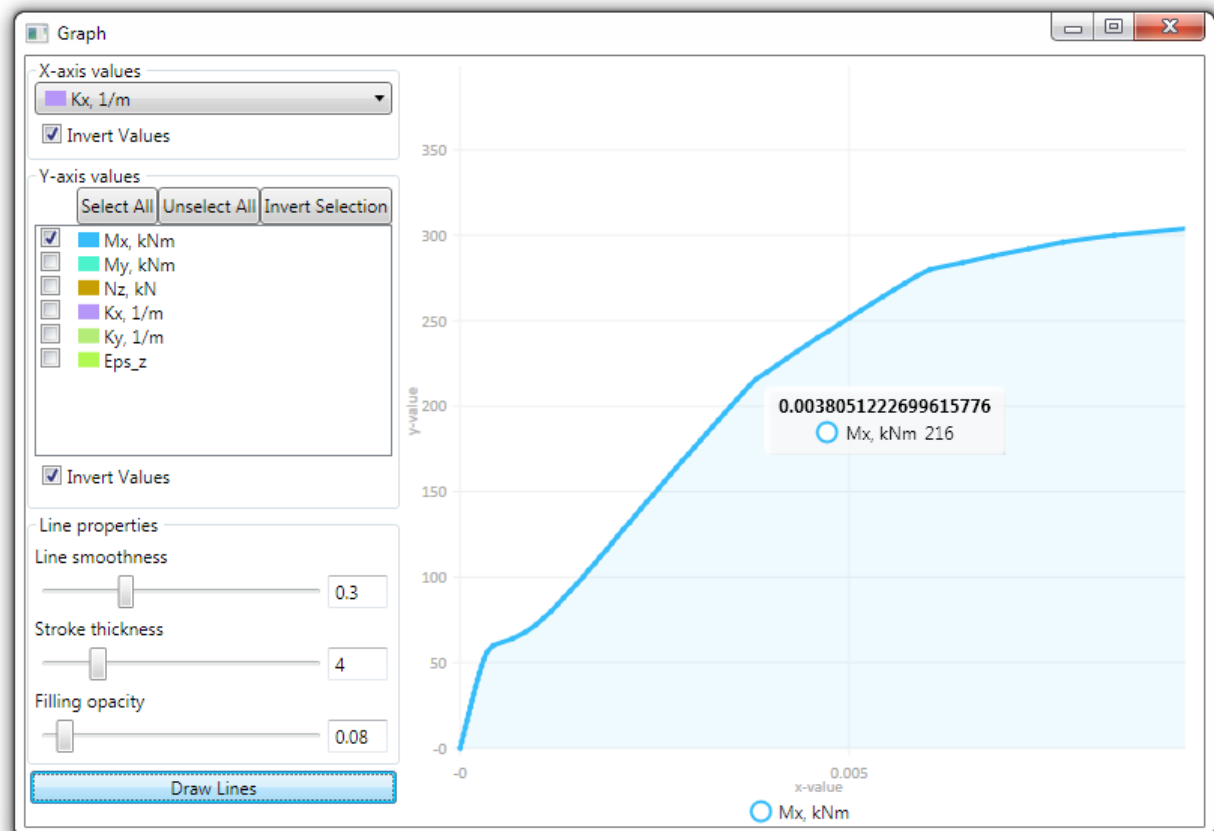
Partial safety factors

Material Safety Factors			
Take	Name	Description	Add
<input checked="" type="checkbox"/>	Gamma_b1	Coefficient for considering long term calculations	Delete
<input type="checkbox"/>	Gamma_b2	Coefficient for plain concrete structures	Copy
<input type="checkbox"/>	Gamma_b3	Coefficient for considering bleeding in vertical placem	Partial frs

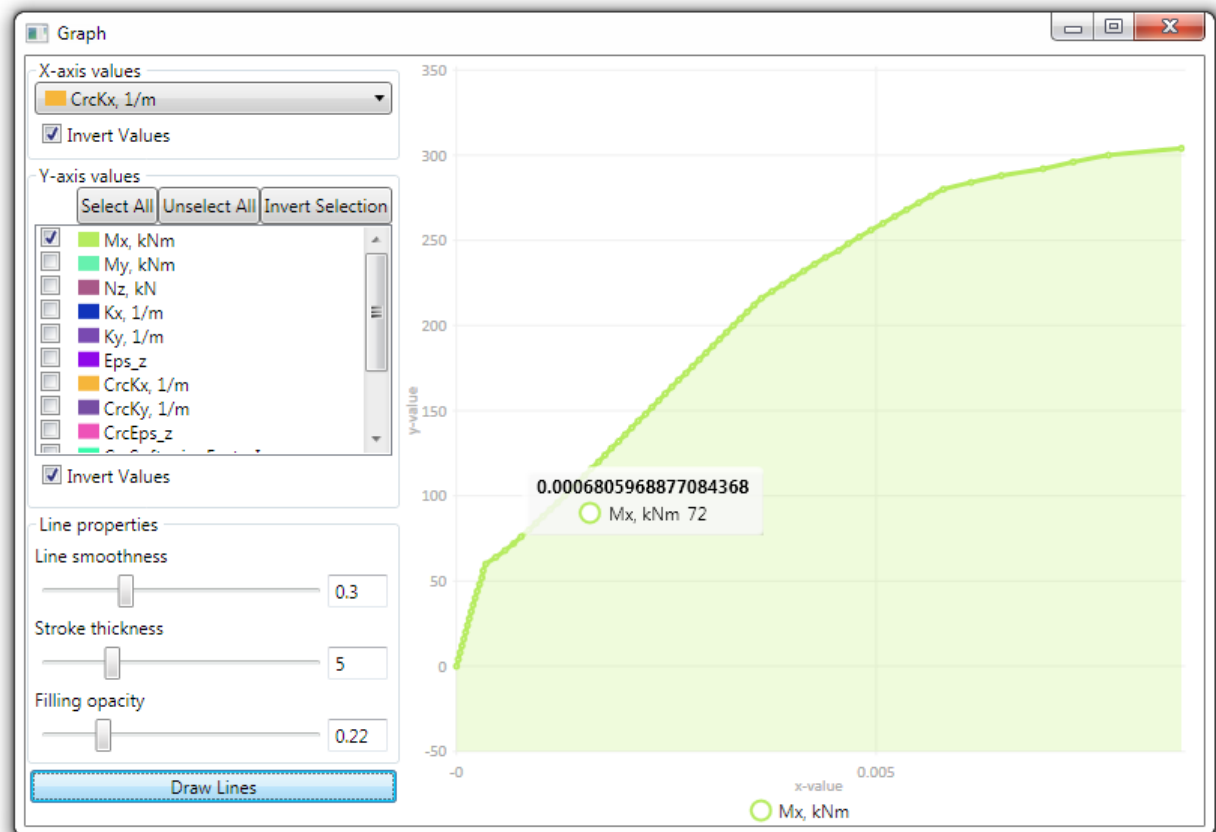


Behavior diagrams

Moment-curvature diagram



Typical moment-curvature diagram for reinforced concrete cross-section (for cross-section of crack)



Typical moment-curvature diagram for reinforced concrete cross-section (average between cracks)

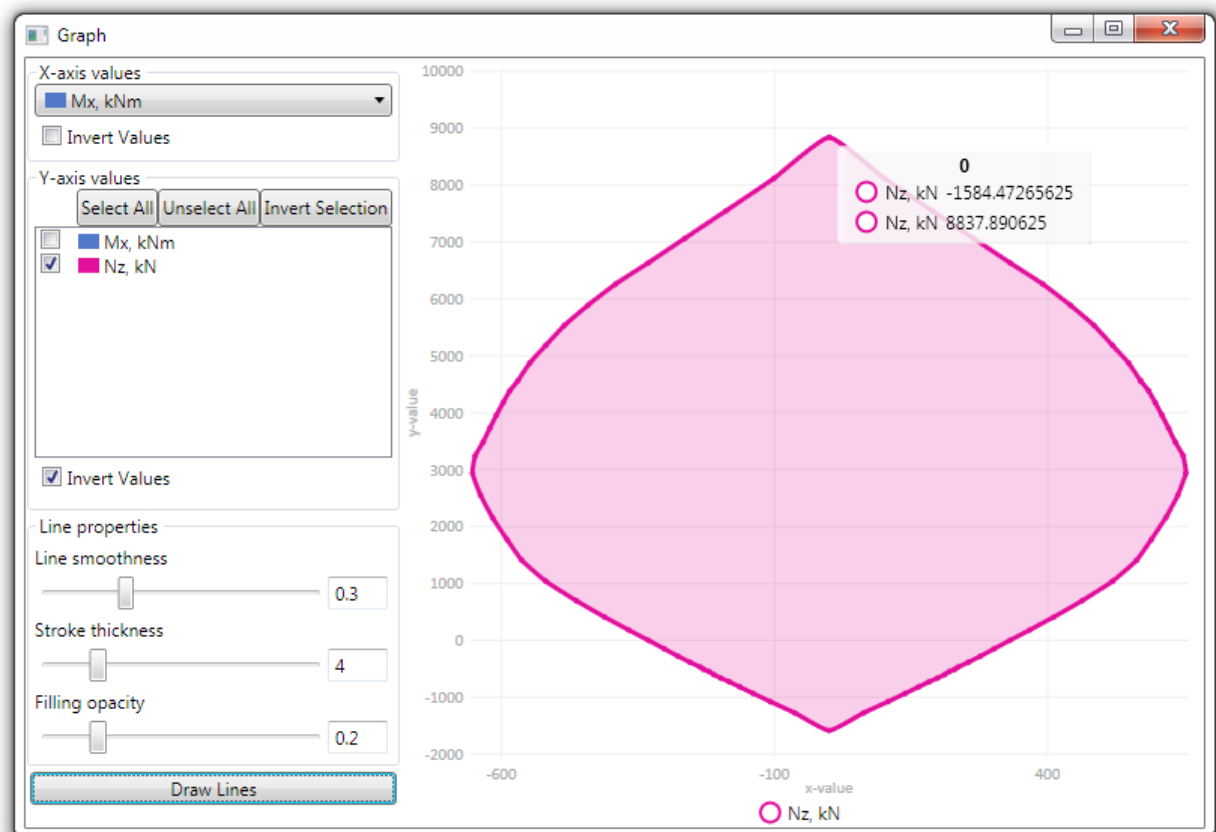
Interaction diagram

The 'Diagram properties' dialog box contains the following settings:

Property	Value	Multiplier
Maximum axial force	10000 kN	*-1 *0 *0.5 *2.0
Minimum axial force	-10000 kN	*-1 *0 *0.5 *2.0
Maximum bending moment Mx	10000 kNm	*-1 *0 *0.5 *2.0
Minimum bending moment Mx	-10000 kNm	*-1 *0 *0.5 *2.0
Minimum bending moment My	-0 kNm	*-1 *0 *0.5 *2.0
Point count	80	*-1 *0 *0.5 *2.0

Properties of interaction diagram

Note, points of diagram area uniformly distributed for assigned range of forces, for that reason try to assign minimum and maximum forces (both axial force and bending moment) closely to maximum bearing capacity, as far as possible.



Typical interaction diagram (for N-Mx axis) for reinforced concrete cross-section