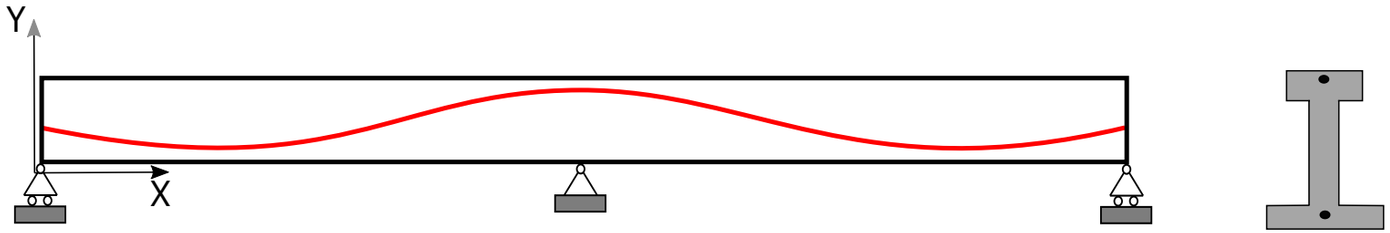
Tutorial

Bond-Slip of Post-Tensioned Reinforcement in 2D Elements



DIANA

DIANA

Outline

1 Description 3
- 1.1 Modeling Approach 4
- 1.2 Material Properties 5

2 Finite Element Model 7
- 2.1 Geometry 10
  - 2.1.1 Beam 10
  - 2.1.2 Reinforcement 12
  - 2.1.3 Support Plates 14
- 2.2 Properties 18
  - 2.2.1 Concrete Beam 18
  - 2.2.2 Tendon 21
  - 2.2.3 Support Plates 25
- 2.3 Boundary Conditions 26
- 2.4 Loads 27
  - 2.4.1 Gravity Load 27
  - 2.4.2 Post-Tensioning Load 28
  - 2.4.3 Concentrated Load 29
  - 2.4.4 Load Combinations 30
- 2.5 Mesh 31

3 Phased Nonlinear Static Analysis 33
- 3.1 Commands 34
  - 3.1.1 Phase 1: Activate Post-Tensioning Loads 34
  - 3.1.2 Phase 1: Structural Nonlinear Analysis 35
  - 3.1.3 Phase 2: Activate Locking of Tendon 36
  - 3.1.4 Phase 2: Structural Nonlinear Analysis - Start Steps 37
  - 3.1.5 Phase 2 - Structural Nonlinear Analysis - Load Steps 38
- 3.2 Results 39
  - 3.2.1 Vertical Displacements 39
  - 3.2.2 Reinforcement Stresses 40
  - 3.2.3 Shear Slip along Reinforcement 41
  - 3.2.4 Shear Stress in the Bond-Slip Interface 42

Appendix A Additional Information 43

1 Description

In this tutorial we analyze a concrete beam with post-tensioned reinforcement (tendon). The beam has a total length of 60 m consisting of two 30 m spans [Fig. 1] with a non-symmetric (on X axis) I-shaped cross-section. The tendon has a curved profile and is post-tensioned by both ends, simultaneously. The phenomena developed during the post-tensioning process are the following:

1. The friction developed between the tendon and the duct leads to losses of post-tensioning force along the tendon
2. The jacks apply a counterbalance force on the anchors. This force is equal to the post-tensioning force and acting in the opposite direction. Therefore, shortening of the beam occurs, which also adds to the post-tensioning losses
3. After the allowable stress limit is reached, the jacks are removed and the tendon is locked at the anchors and grouted

In DIANA there are two solutions for modeling post-tensioned reinforcement in plane stress:

- Post-tensioning of bond-slip reinforcement and the anchors
- Post-tensioning load on embedded (no-bond) reinforcement

This tutorial presented the first solution, since we attempt to realistically model the phases of post-tensioning. Therefore we consider the bond-slip interface failure model and the reinforcement anchor model.

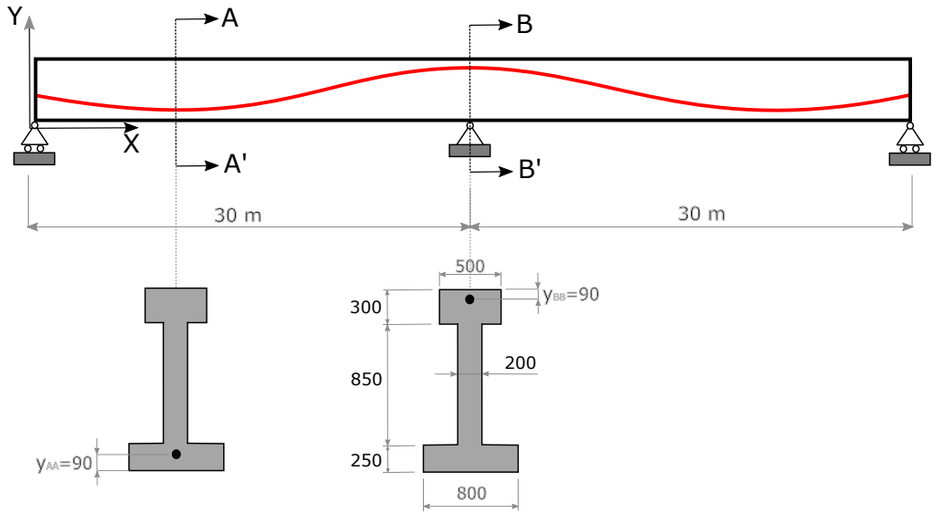


Figure 1: Geometry of the post-tensioned beam

1.1 Modeling Approach

The model has the following characteristics:

- the slipping of the tendon in the concrete is considered by the *truss bond-slip bar reinforcements* in the model. The reinforcement bar is modeled with bond-slip truss elements. The bond-slip behavior is controlled by interface elements that are automatically generated around the reinforcement bar. Note that assigning bond-slip reinforcement in DIANA is similar to modeling of embedded reinforcements, except from:
  - Assigning bond-slip material properties. In this tutorial:

    The relation between the normal traction and the normal relative displacement is assumed to be linear elastic.

    The relation between the shear traction and the slip is assumed as a nonlinear function. For this relation five predefined curves are offered in DIANA along with a user-defined multi-linear curve. Here, we use the *Cubic function according to Doerr* [Fig. 2].
- The concrete beam is modeled with quadrilateral 2D plane stress elements using linear interpolation.
- The concrete material is assumed linear elastic
- The beam is constrained vertically at both ends and at mid span. A horizontal constraint is also applied at mid span. Elastic steel plates are used to model the supports.
- For the post-tensioning of the bond-slip reinforcement we apply a *phased nonlinear static* analysis:

  Phase 1:
  - The tendon is jacked at both ends simultaneously. Weak bonding is defined between tendon and concrete since the tendon is still not grouted.

  Phase 2
  - The tendon is now grouted. The weak bond interface properties are changed into stiff elastic properties. The prestress loads are removed.
  - A local vertical load is applied at the middle of the left span.

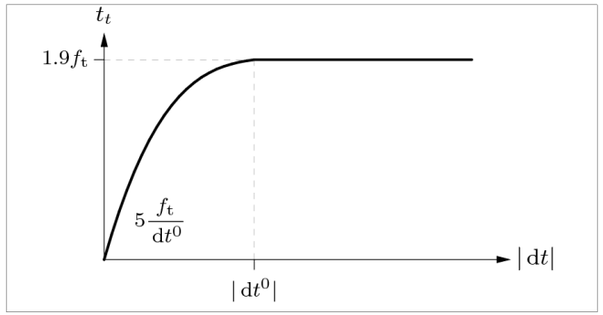


Figure 2: Cubic function by Doerr

1.2 Material Properties

The material properties used in the model are defined in Table 1. For the concrete we use the fib Model Code for Concrete Structures 2010. For the bond-slip reinforcement we define different material properties for the weak nonlinear bond (*Phase 1*) and the stiff nonlinear bond (*Phase 2*). We use a very low shear stiffness for the bond-slip reinforcement interface in combination with a high value for *shear slip at start plateau* in the weak bond phase. Due to grouting we use a high shear stiffness for the bond-slip reinforcement interface in phase 2. In this phase the *shear slip at start plateau* is equal to 0.1 m. We include reinforcement anchor properties at the begin and end of the bar with high stiffnesses for the springs on anchor direction and on normal direction to anchor and no failure model.

Table 1: Material properties

Concrete			
Model code		fib Model Code 2010	
Grade		C55	
Reinforcement steel		(elastic)	
Young's modulus	E_x	2.1e+11	N/m^2
Poisson's ratio	ν	0.3	
Mass density	ρ	7.85E+3	kg/m^3
Bond-slip reinforcement interface			
*Weak bond (*empty duct - Phase 1*)*			
Normal stiffness modulus	D_{11}	1.0E+12	N/m^2
Shear stiffness modulus	D_{22}	2.0E+06	N/m^2
Failure model		cubic function by Doerr	
Parameter c		20	N/m^2
Shear slip at start plateau		10.0	m
Stiff bond (grouted - Phase 2*)*			
Normal stiffness modulus	D_{11}	1.0E+12	N/m^2
Shear stiffness modulus	D_{22}	2.0E+10	N/m^2
Failure model		cubic function by Doer	
Parameter c		20	N/m^2
Shear slip at start plateau		0.1	m
Bond-slip reinforcement anchor			
Weak and stiff bond			
Spring stiffness in anchor direction	K_n	1.0E+12	N/m
Spring stiffness normal to anchor	K_s	1.0E+12	N/m

2 Finite Element Model

For the modeling session we start a new project for a two dimensional structural analysis.

DIANAIE

Main menu → File → New [Fig. 3]

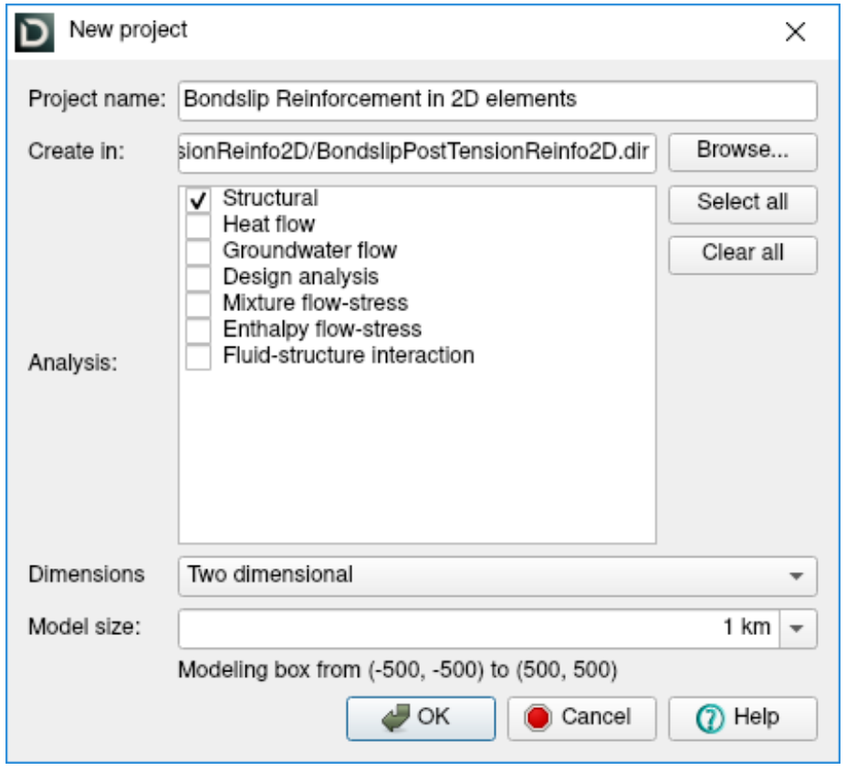


Figure 3: New project dialog

We choose meter for the unit length, newton for force and kilograms for mass. The units and the directions are displayed in the reference system section of the geometry browser.

DIANAIE

Geometry browser → Reference system → Units [Fig. 4]
Property Panel [Fig. 5]

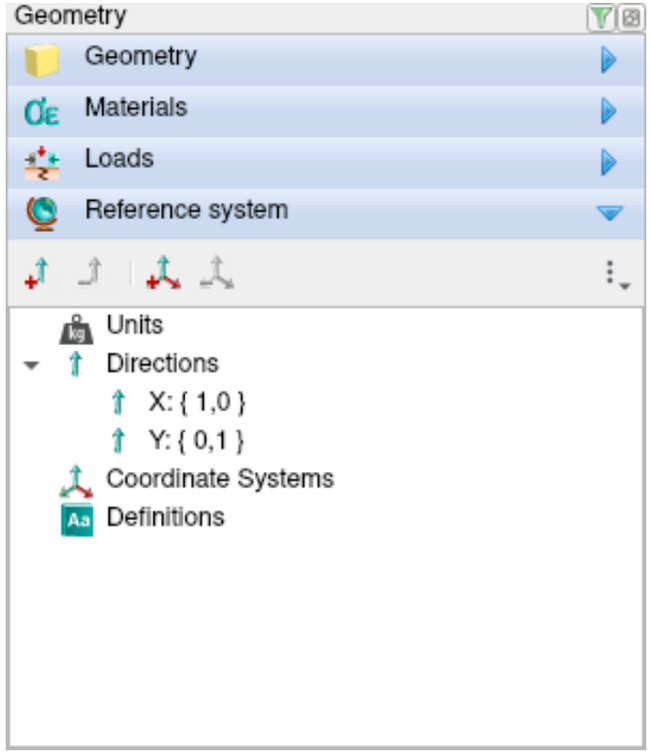


Figure 4: Geometry browser - units

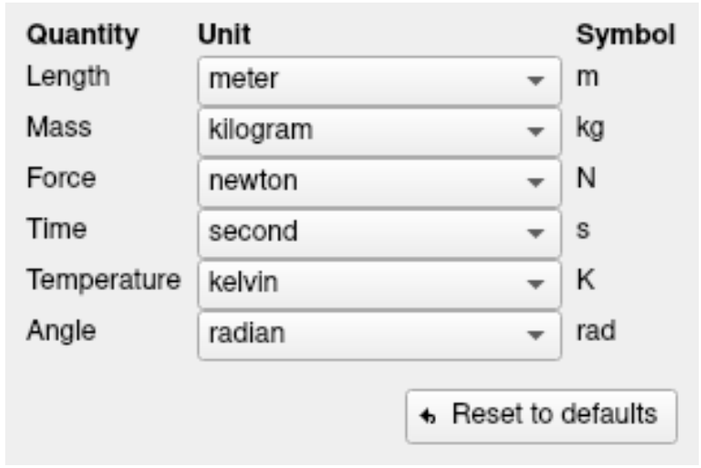


Figure 5: Properties panel - units

In the mesher preferences we define linear mesh.

Main menu → Mesh → Mesher preferences [Fig. 6]

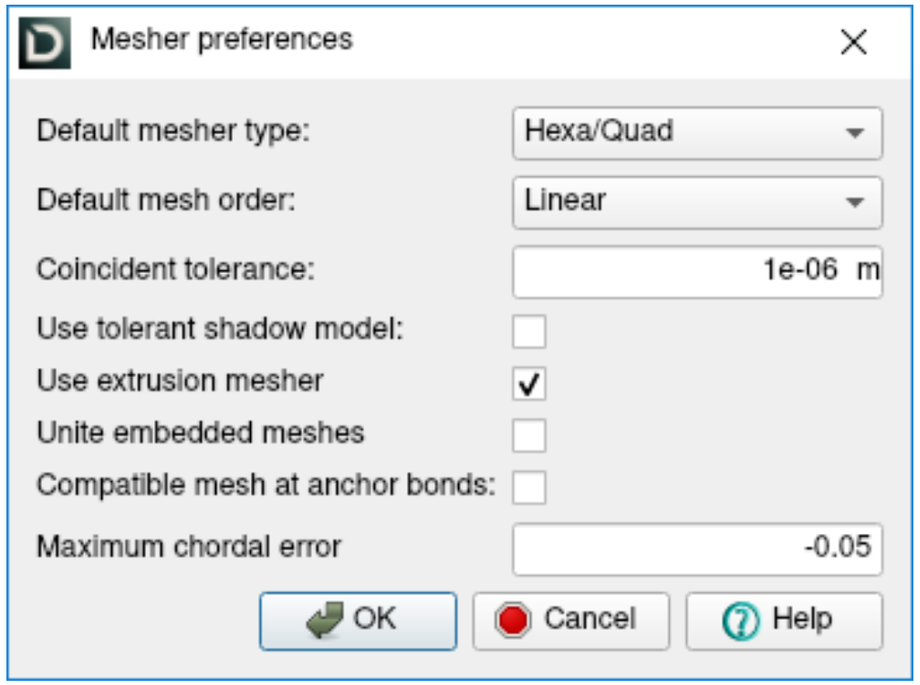


Figure 6: Mesher preferences

2.1 Geometry

2.1.1 Beam

The cross-section of the I-beam consists of three parts, namely: bottom flange, web and top flange. We draw these shapes as rectangular sheets with dimensions 60 $\times$ 0.25 m, 60 $\times$ 0.85 m and 60 $\times$ 0.30 m, respectively.

DIANAIE

Main menu → Geometry → Create → Polygon sheet [Fig. 7] [Fig. 8] [Fig. 9]

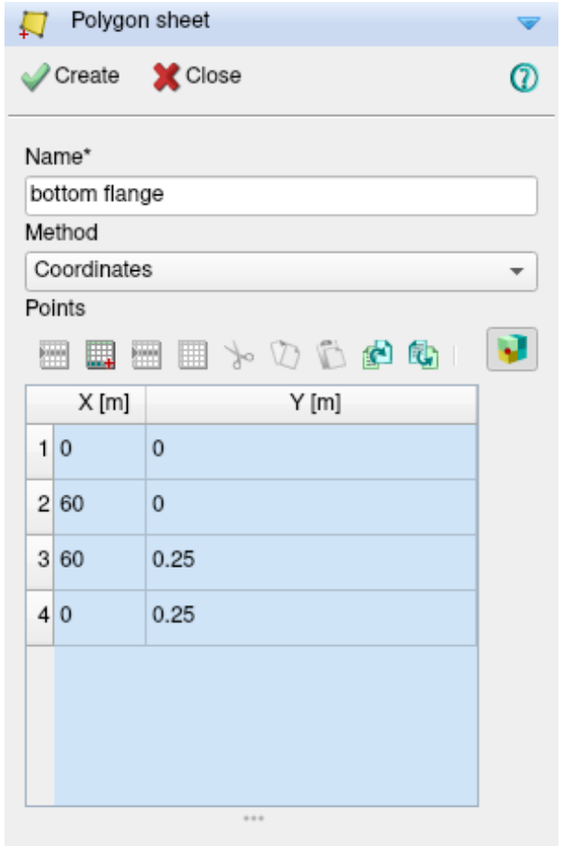


Figure 7: Add polygon sheet - bottom flange

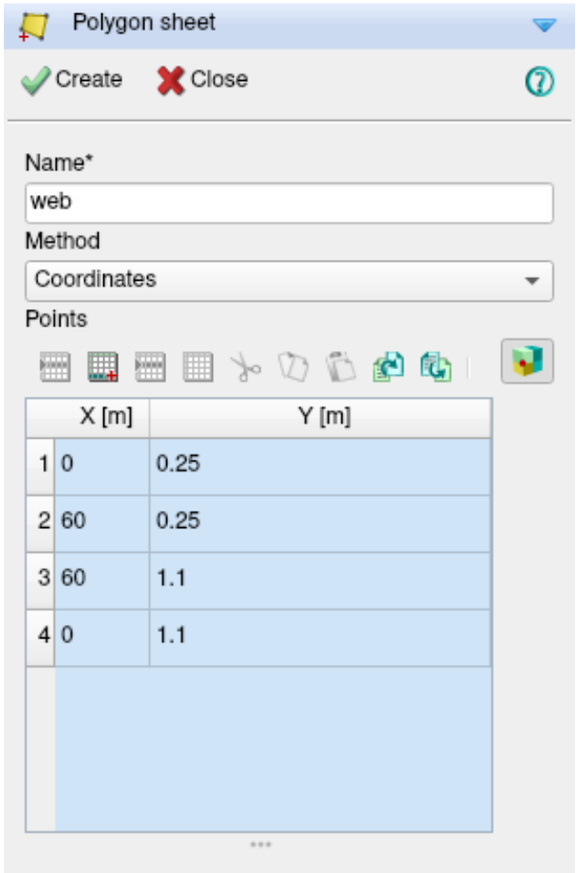


Figure 8: Add polygon sheet - web

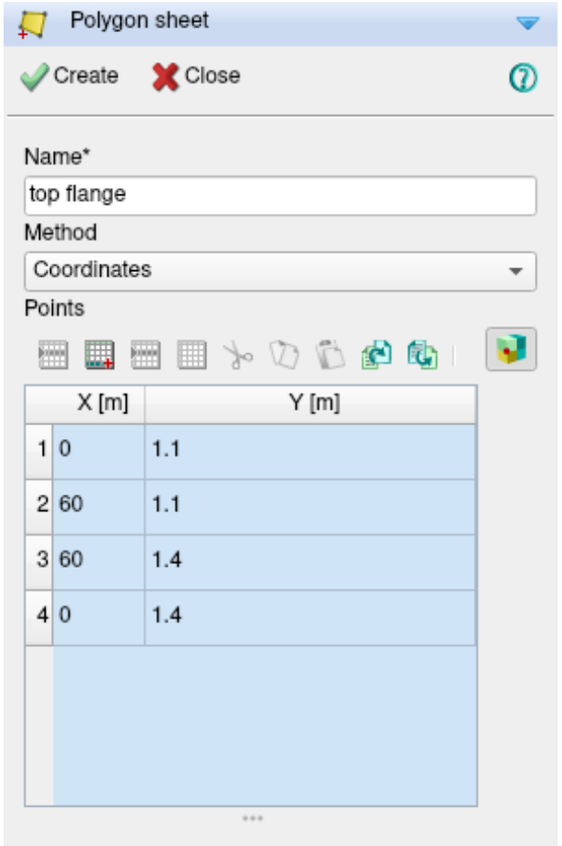


Figure 9: Add polygon sheet - top flange

Figure 10: Geometry - bottom flange, web and top flange

2.1.2 Reinforcement

We draw the curved profile of the tendon.

DIANAIE

Main menu → Geometry → Create → Curve [Fig. 11] [Fig. 12]

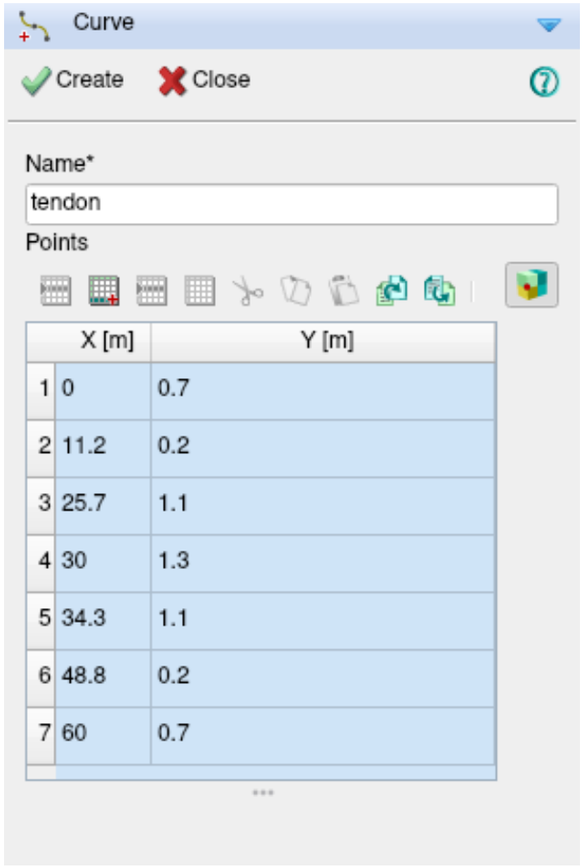


Figure 11: Add curve - tendon (reinforcement)

Note: in cases that the tendon profile exhibits larger changes of curvature it might be required to define more points of the curved line to acquire a smoother profile.

Figure 12: Geometry - concrete beam and reinforcement (tendon) profile

2.1.3 Support Plates

Next, we create the geometry of the support plates (1.2 × 0.30 m). A vertex is created and projected at the bottom edge of each plate. These vertices are used to attach the boundary constraints [Section 2.3].

Left support plate
First, we create the left support plate. Then, we add a vertex at the middle of its bottom edge. We project this vertex on the plate and remove it (by not activating the option *Keep tools*). The remaining two plates are created by mirroring and copying of the left support plate.

DIANAIE

Main menu → Geometry → Create → Polygon sheet [Fig. 13] [Fig. 16]
Main menu → Geometry → Create → Point [Fig. 14] [Fig. 16]
Main menu → Geometry → Modify → Projection [Fig. 16] [Fig. 15]

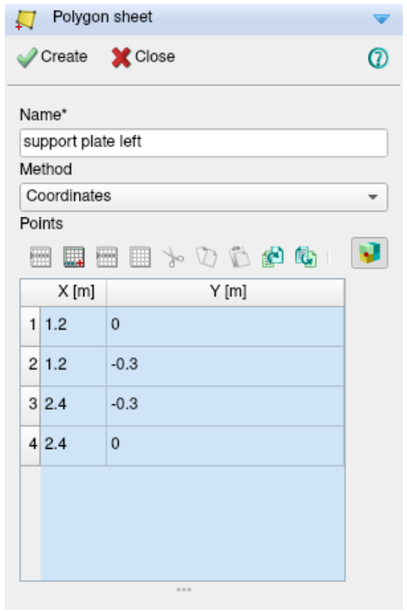


Figure 13: Add polygon sheet: left support

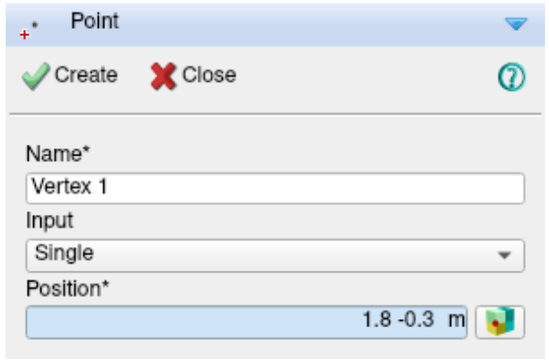


Figure 14: Add vertex

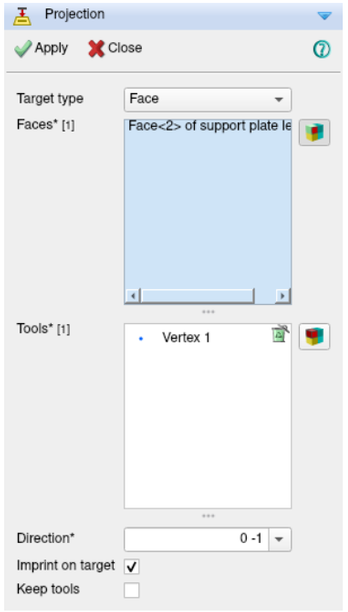


Figure 15: Project vertex on suppport edge

Note that we use different colors for different shapes.

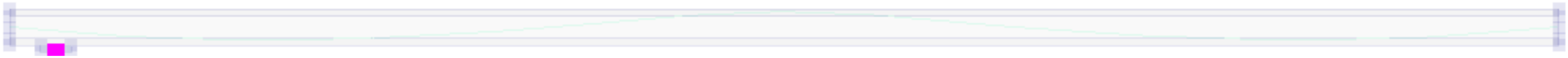


Figure 16: Left support: add and project vertex

Middle & Right Support Plates
We create the right support plate by mirroring the left support plate. The selected pivot is the bottom middle point of the beam. We also activate the option *Keep original*; otherwise, the left support plate is removed. The easiest way to create the middle support is by copying and moving the left support. This is done by use of the *Array copy* geometry tool. Finally, we rename the two new shapes.

DIANAIE

Main menu → Geometry → Modify → Mirror [Fig. 17] [Fig. 20] [Fig. 21]
Main menu → Geometry → Modify → Array copy [Fig. 18] [Fig. 19] [Fig. 21]
Geometry browser → Geometry → Shapes → Shape 1 → Rename → Support plate right
Geometry browser → Geometry → Shapes → Shape 2 → Rename → Support plate middle

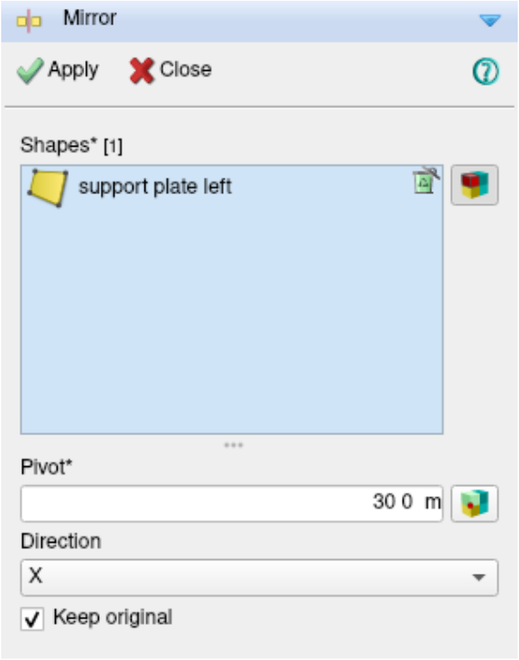


Figure 17: Mirror shape - left support

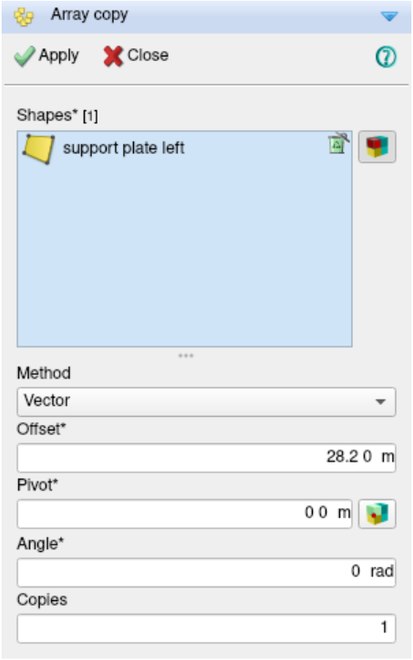


Figure 18: Array copy - left support

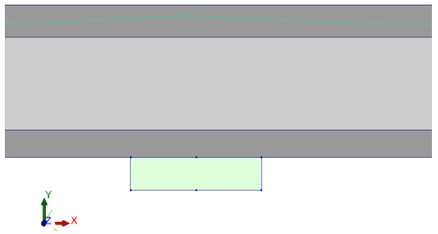


Figure 19: Middle support plate

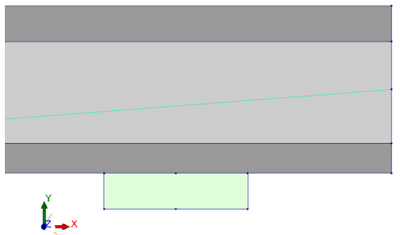


Figure 20: Right support plate

Figure 21: Geometry: beam, tendon and support plates

2.2 Properties

The next step is to assign properties to the geometric shapes.

2.2.1 Concrete Beam

The concrete beam is modeled with membrane elements. The properties of its parts (bottom flange, web, top flange) are assigned separately due to different thickness.

Bottom Flange
We assign the concrete material to the bottom flange and define its properties [Table 1].

DIANAIE

Main menu → Geometry → Assign → Shape properties [Fig. 22]
Shape properties → Material → Add material [Fig. 23] → Edit material [Fig. 24]
Shape properties → Geometry → Add new geometry [Fig. 25]

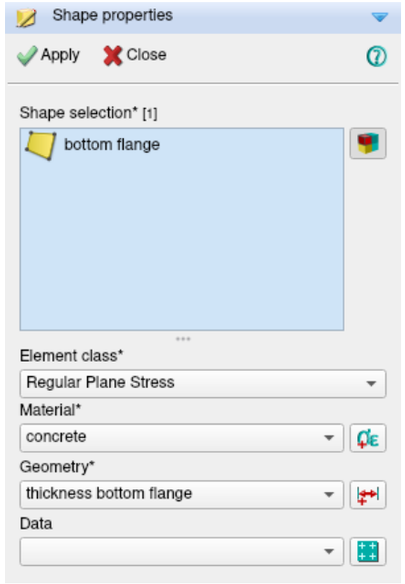


Figure 22: Property assignments - bottom flange

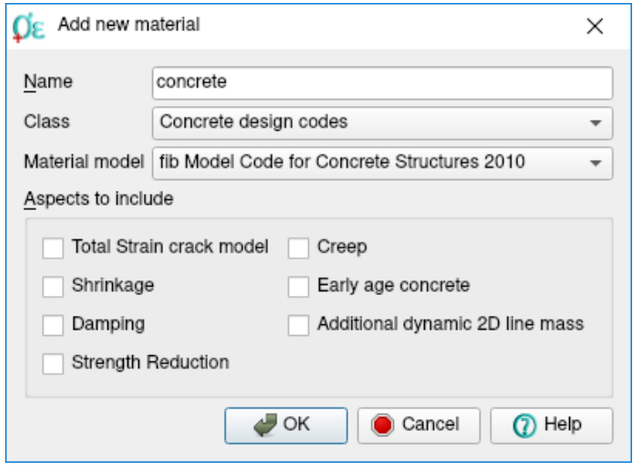


Figure 23: Add material - concrete

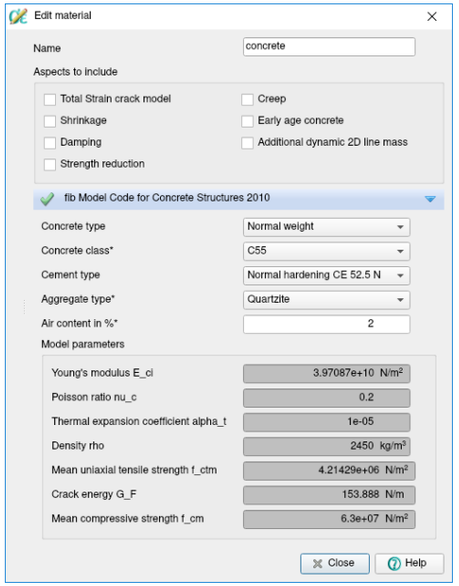


Figure 24: Edit material - boncrete

We define the element geometry with thickness equal to the bottom flange width (0,8 m).

DianaIE

Shape properties → Geometry → Add new geometry [Fig. 25]

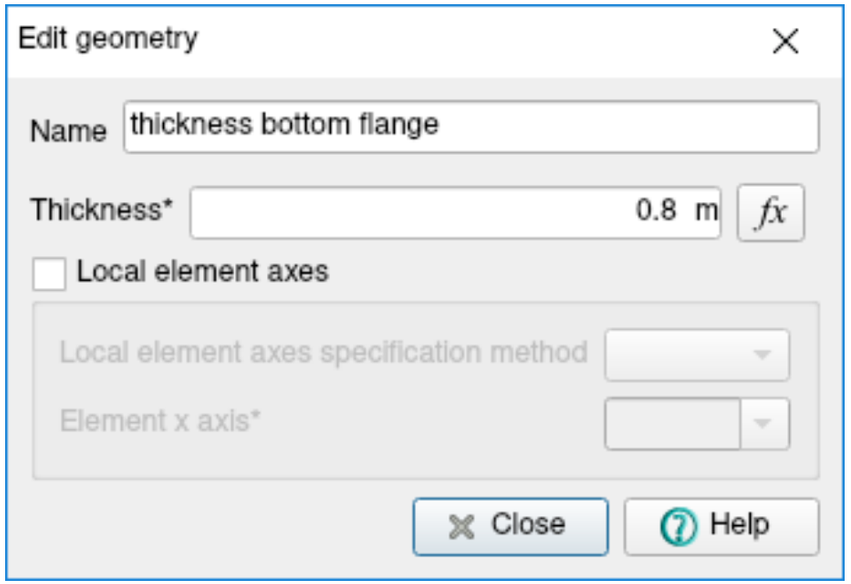


Figure 25: Edit geometry - bottom flange

Web and Top Flange
Similarly, we assign properties to the web and top flange. Their thickness is equal to the web width (0.2 m) and the top flange width (0.5 m), respectively. Concrete is selected as material (its properties were already defined [Fig. 23 to 24]). We may set the local axis (default X axis).

Main menu → Geometry → Assign → Shape properties [Fig. 26] [Fig. 29]
Shape properties → Geometry → Add new geometry [Fig. 27] [Fig. 28]

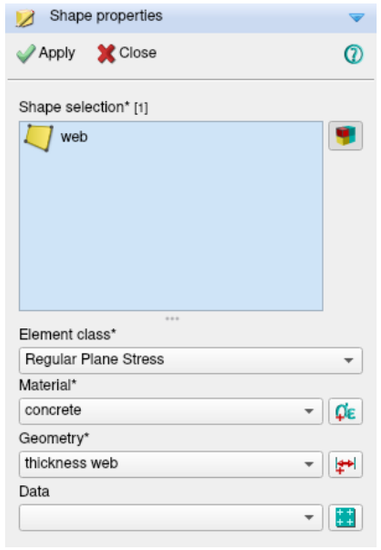


Figure 26: Property assignments - web

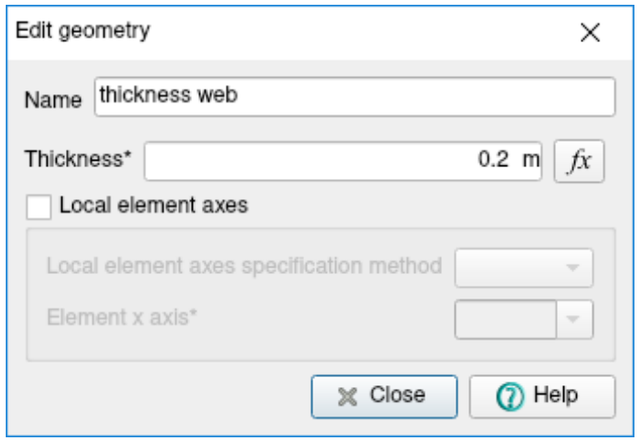


Figure 27: Edit geometry - web

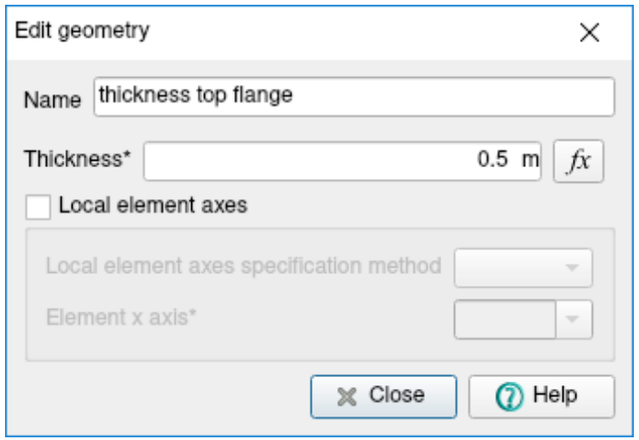


Figure 28: Edit geometry - top flange

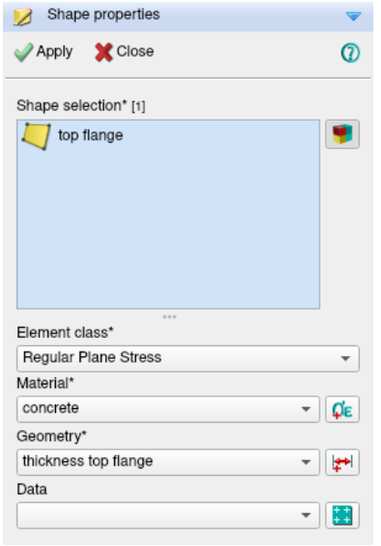


Figure 29: Shape property assignments - top flange

2.2.2 Tendon

We assign bond-slip reinforcement properties to the tendon.

Tendon not grouted
For the type of reinforcement we use truss bond-slip bar. We define the elastic properties of the reinforcement bar and the weak bond properties of the bond-slip interface [Table 1] when the tendon is not grouted (*Phase 1*). We use the option *Reset state parameters on material change*. This option is particularly useful for the simulation of grouting of the post-tensioned cables where soft interface material becomes stiff after grouting without causing any strain and stress increment in the interface.

DIANAIE

<Select the correspondent reinforcement set in the Geometry browser >

Main menu → Geometry → Assign → Reinforcement properties [Fig. 30]

Reinforcement properties → Material → Add material [Fig. 31] → Edit material [Fig. 32] [Fig. 34]

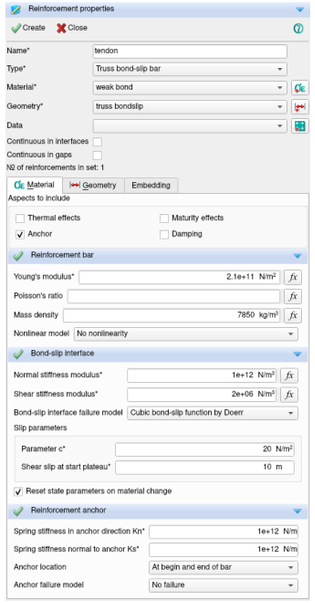


Figure 30: Property assignments - tendon

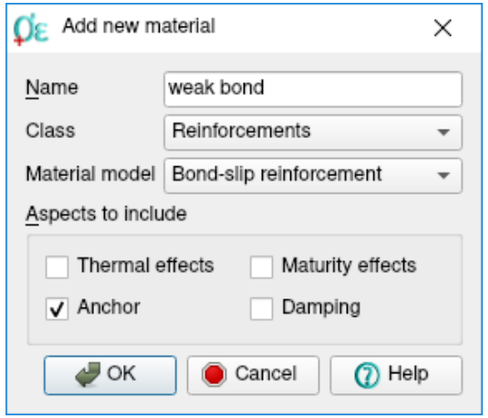


Figure 31: Add material - tendon not grouted

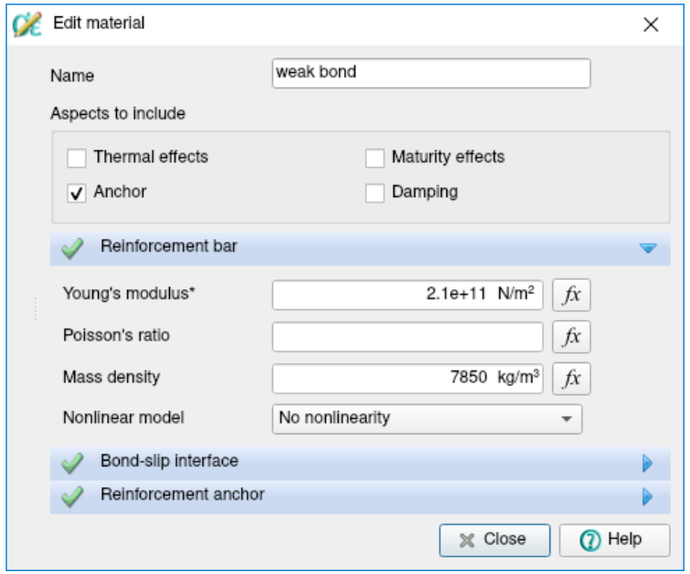


Figure 32: Edit material - tendon not grouted

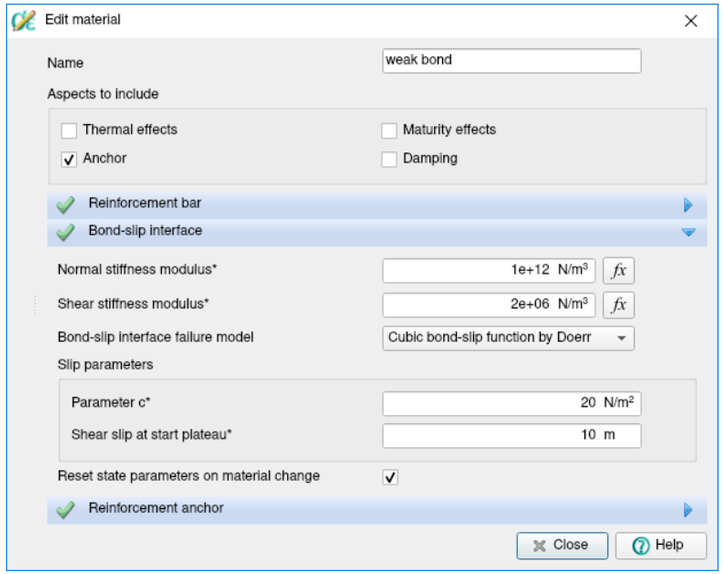


Figure 33: Edit material - tendon not grouted

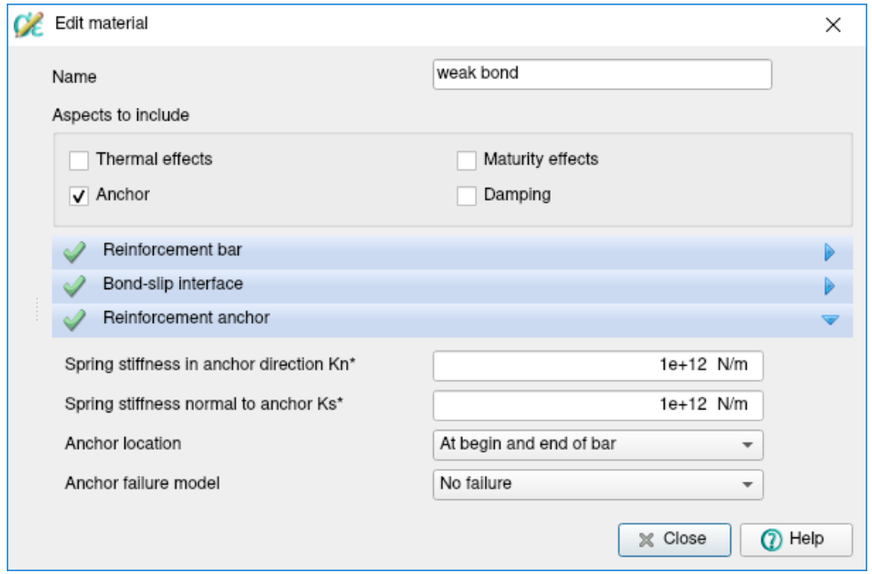


Figure 34: Edit material - anchor

Note that, here, the bar and the concrete are assumed to be in full contact. This is done by not specifying a *contact perimeter* [Fig. 35] for the interface elements; thus, letting DIANA automatically calculate it by using the diameter defined for the bar elements.

DIANAIE

Reinforcement properties → Geometry → Edit geometry [Fig. 35]

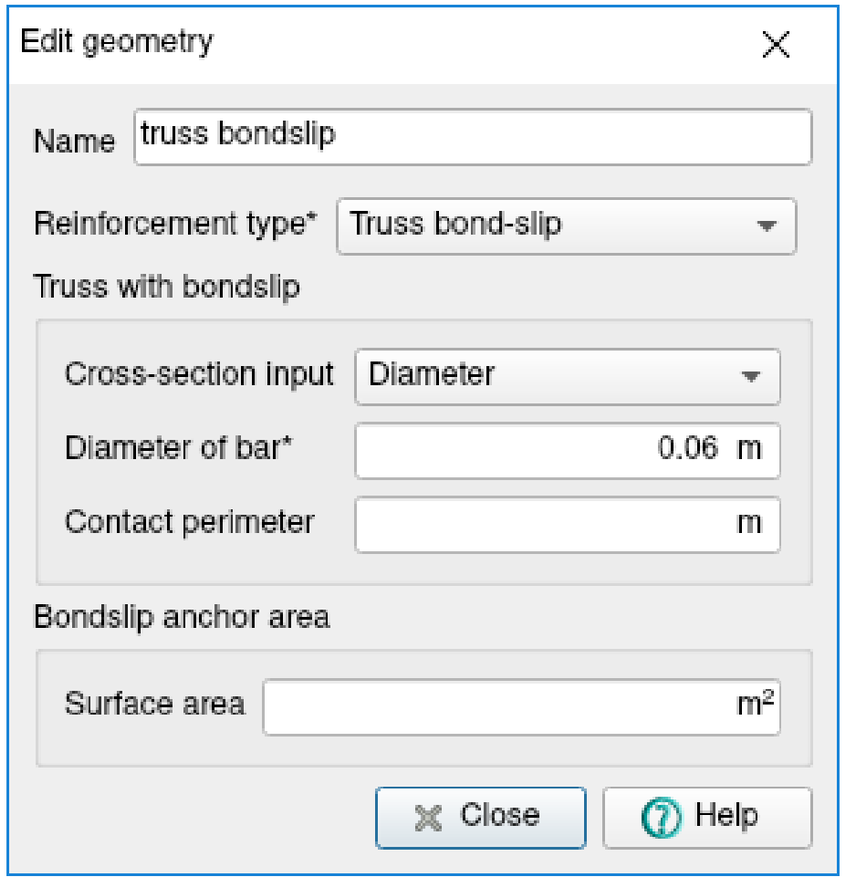


Figure 35: Edit geometry - tendon

Tendon grouted

We define a new material, with stiff bond-slip interface properties [Table 1], to use for the grouted tendon (*Phase 2*). We do this by duplicating and editing the existing *weak bond* material [Fig. 31 to 33]. This assignment is considered in the *Phased* analysis set-up [Section 3]. We change the value of the shear stiffness modulus. We also change the value of the slip at start plateau to a realistic value of 0.1 m.

DIANAIE

Geometry browser → Materials → weak bond → Duplicate [Fig. 36] → Rename → stiff bond [Fig. 37]

Geometry browser → Materials → stiff bond → Edit material [Fig. 38]

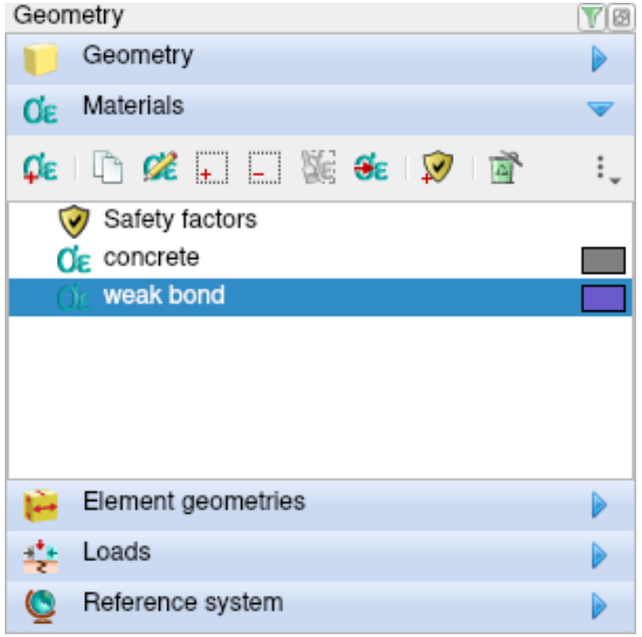


Figure 36: Duplicate material

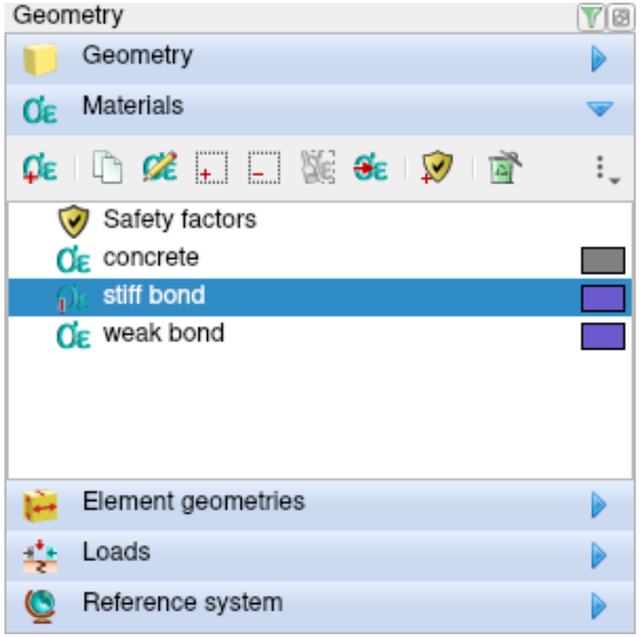


Figure 37: Rename material

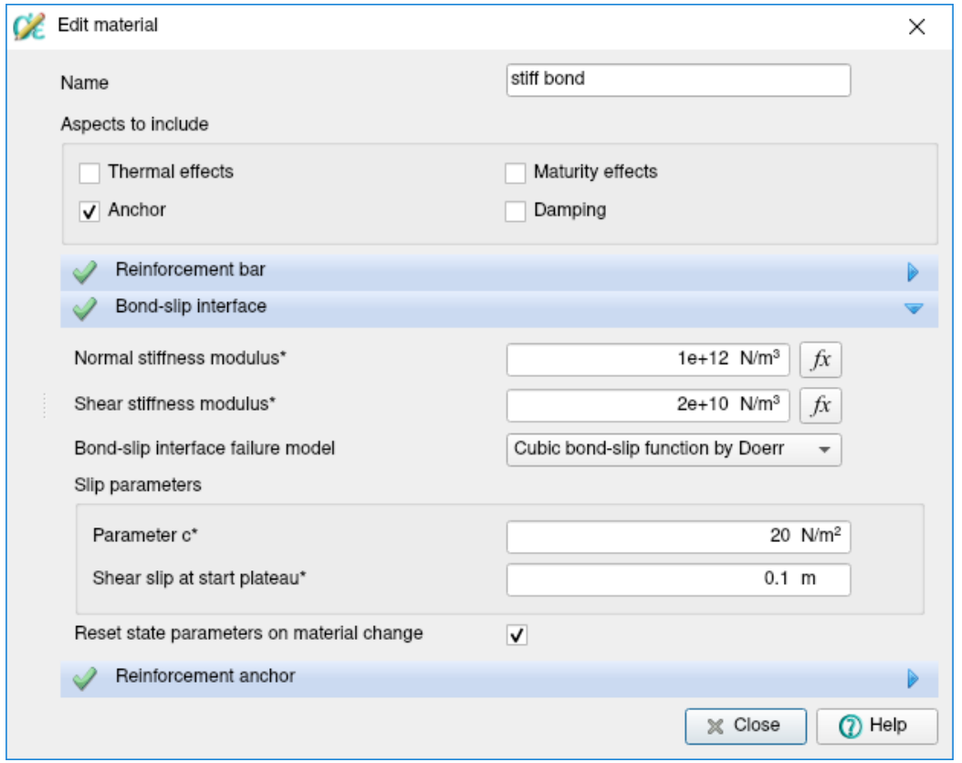


Figure 38: Edit material - tendon grouted

2.2.3 Support Plates

We assign properties to the support plates. For steel we define linear elastic behavior [Table 1]. The thickness is equal to the width of the bottom flange (0.8 m).

DIANAIE

Main menu → Geometry → Assign → Shape properties [Fig. 39]
Shape properties → Material → Add material [Fig. 40] → Edit material [Fig. 41]
Shape properties → Geometry → Add new geometry [Fig. 39] [Fig. 42]

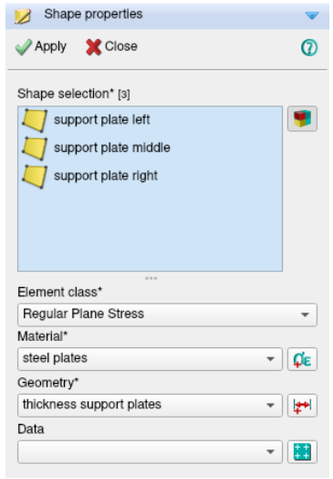


Figure 39: Property assignments - support plates

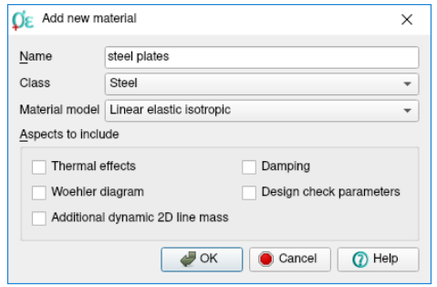


Figure 40: Add material - steel plates

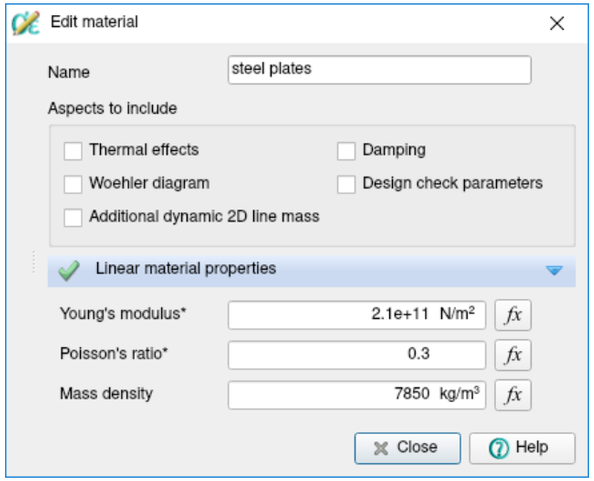


Figure 41: Edit material - steel plates

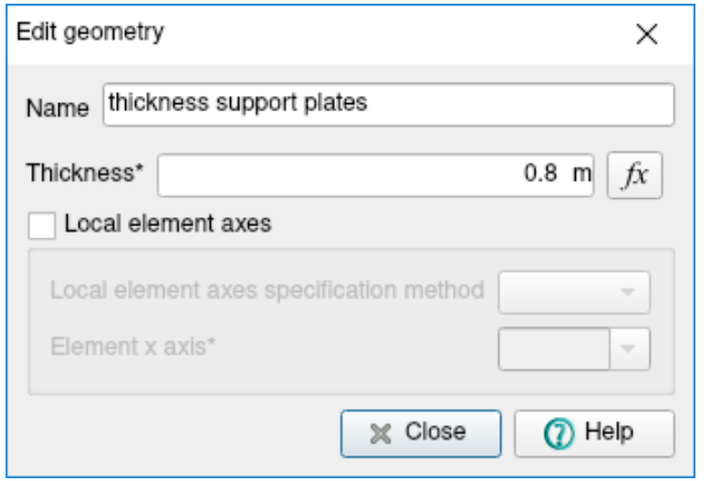


Figure 42: Edit geometry - plates

2.3 Boundary Conditions

We restrict the translation in the Y direction at the middle points (bottom edge) of the three support plates. We restrict the translation in the X direction at the middle point (bottom edge) of the middle support plate.

DIANAIE

Main menu → Geometry → Assign → Supports [Fig. 43] [Fig. 44] [Fig. 45]

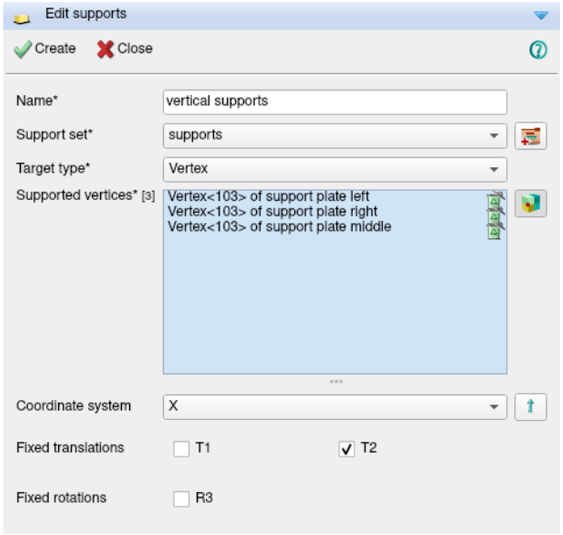


Figure 43: Attach support - vertical support

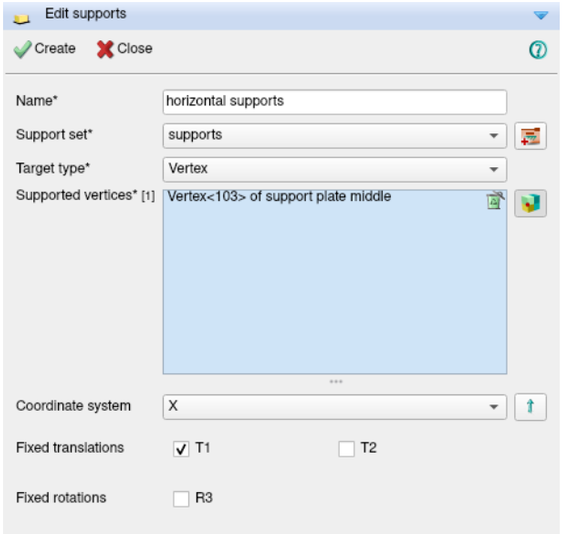


Figure 44: Attach support - horizontal support

Figure 45: Supports - horizontal and vertical

2.4 Loads

2.4.1 Gravity Load

We apply the dead weight as a global load.

DianaIE

Main menu → Geometry → Assign → Global loads [Fig. 46]

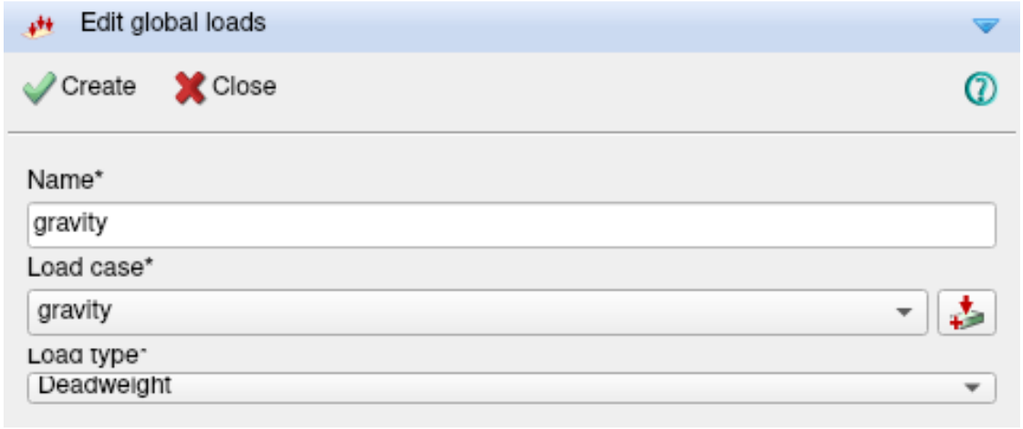


Figure 46: Global load

Main menu → Geometry → Assign → Loads [Fig. 47]

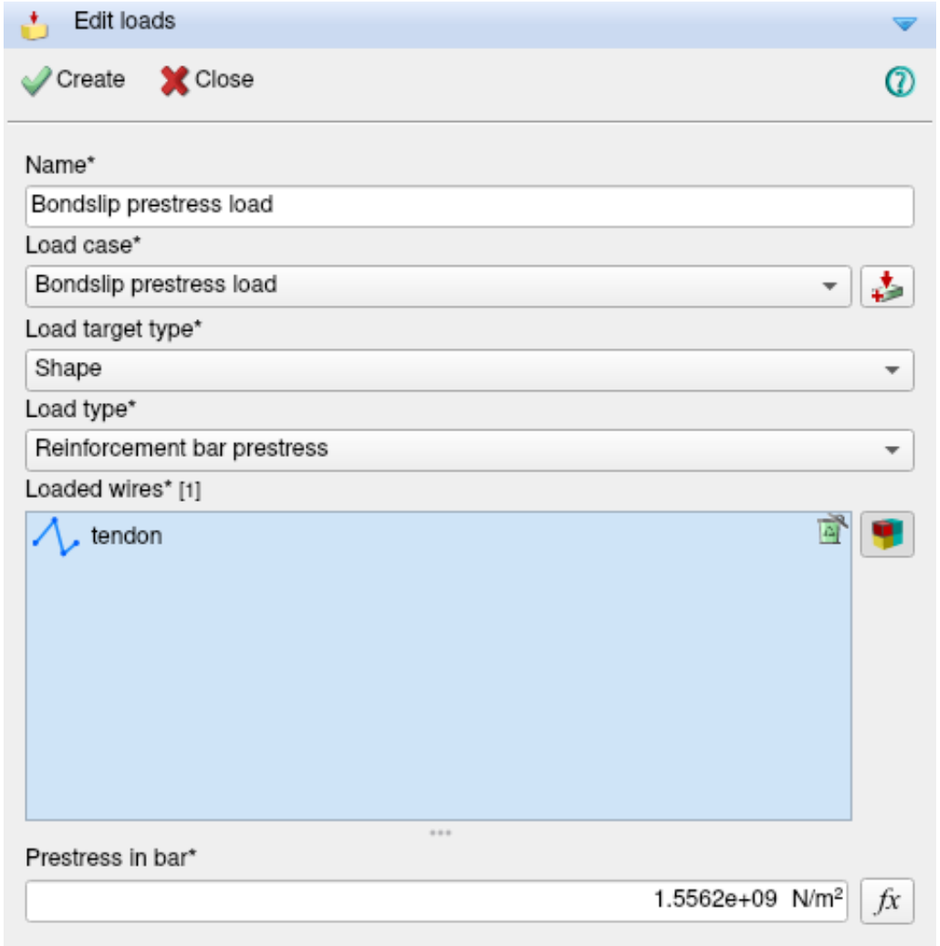


Figure 47: Attach load - reinforcement bar prestress

2.4.3 Concentrated Load

A concentrated force is applied at the left middle span. To do this,we first add and project a vertex [Fig. 49 to 50] on which we attach the point load [Fig. 51] of 1000 kN. This load acts after the tendon is grouted. We assign it to a new load case named ”point load” . Note that, during the projection, we do not activate the *Keep tools* option so that, afterwards, the vertex is automatically removed.

DIANAIE

Main menu → Geometry → Create → Point [Fig. 49] [Fig. 48]

Main menu → Geometry → Modify → Projection [Fig. 50] [Fig. 48]

Main menu → Geometry → Assign → Loads [Fig. 51] [Fig. 48]

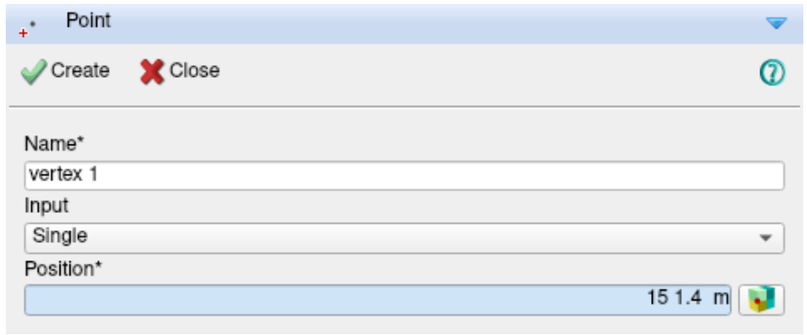


Figure 49: Add vertex (left span)

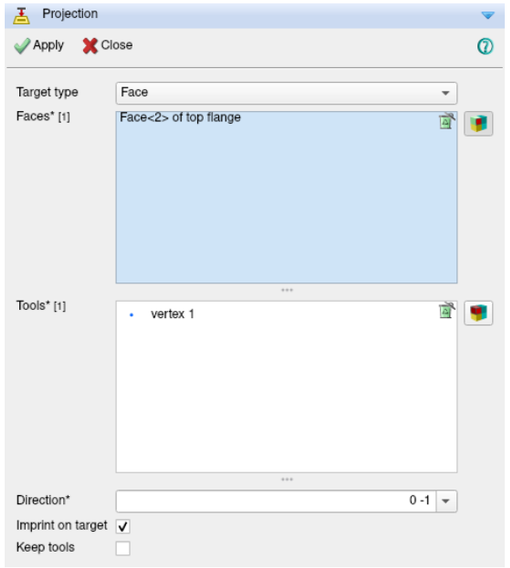


Figure 50: Projection of vertex

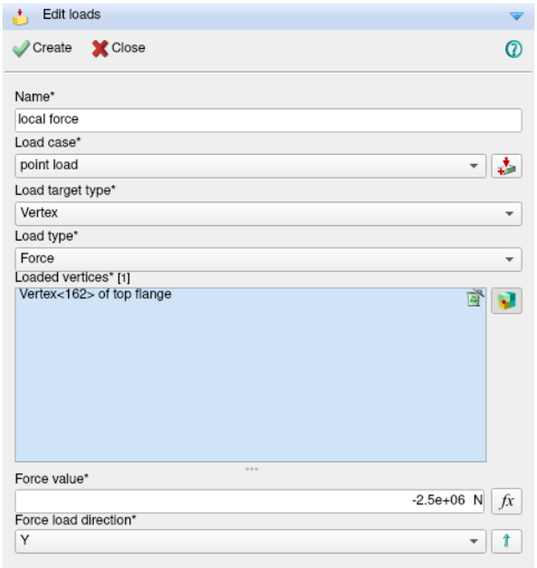


Figure 51: Attach point load

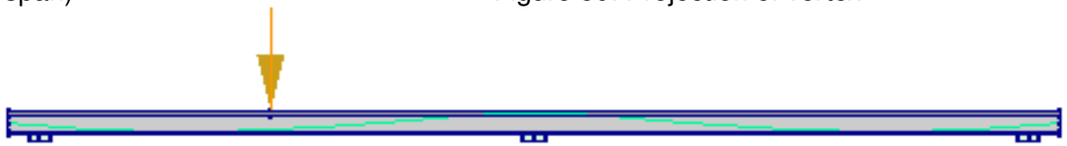


Figure 48: Local (point) load

2.4.4 Load Combinations

The defined load cases of gravity and post-tensioning loads act simultaneously during the 1^{st} phase of the analysis. Therefore, we define a new combination including the specific load cases (each with factor of 1) in the *geometry load combination table* [Fig. 53].
Note that load combinations for each load case individually (3 in total) are automatically defined once the *geometry load combination table* is activated. Also note that, once the *geometry load combination table* is active, we must use the defined load combinations for the analysis, instead of the existing load cases; even if the load combination includes only one load case.

DIANAIE

Main menu → Geometry → Loads → Open geometry load combinations table [Fig. 52]

Geometry Load Combinations → Add geometry load combination → Geometry Geometry load combination 4 [Fig. 53]

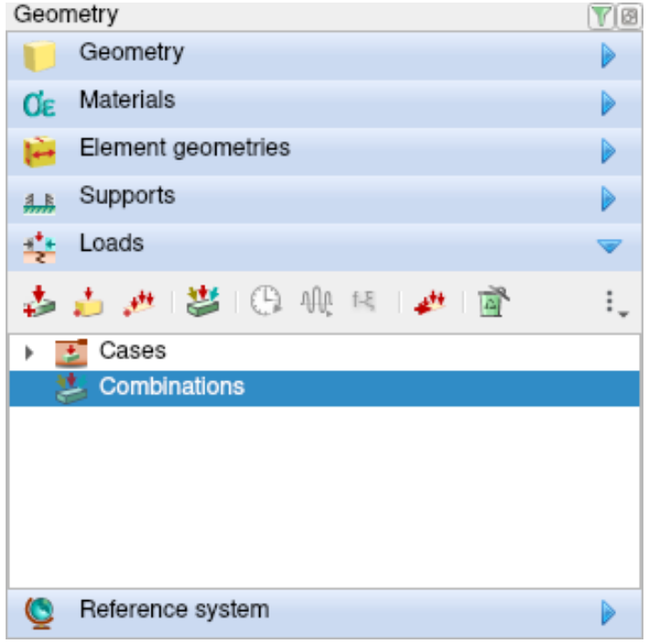


Figure 52: Geometry load combinations

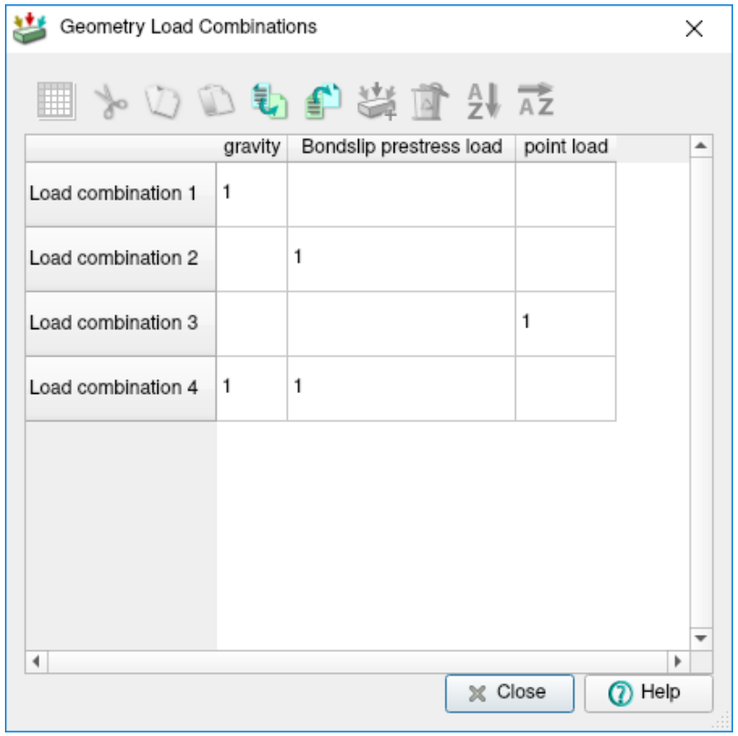


Figure 53: Geometry load combinations table

2.5 Mesh

We set the size of the membrane elements to 0.15 m. The quadrilateral mesher type was already defined in the project settings [Fig. 3]. We generate the mesh and activate the *Shrunken shading* view.

DIANAIE

Main menu → Geometry → Mesh → Mesh properties [Fig. 54]

Main menu → Geometry → Mesh → Generate mesh [Fig. 56]

Mesh browser → Shrunken shading with feature edges [Fig. 55] – [Fig. 57]

< *The mesh view options are active when you select the mesh components in the mesh view* >

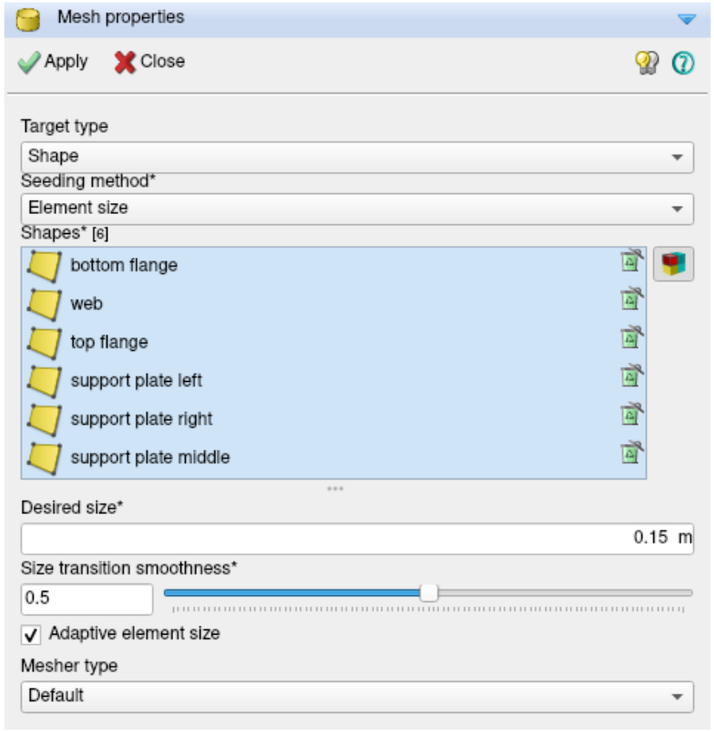


Figure 54: Set mesh properties

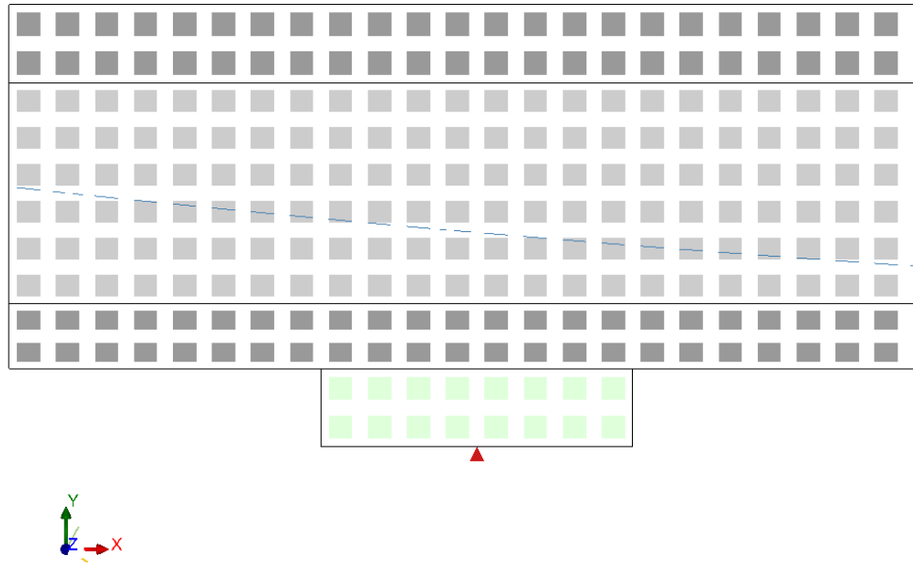


Figure 55: Mesh at left side (anchor reaction force)

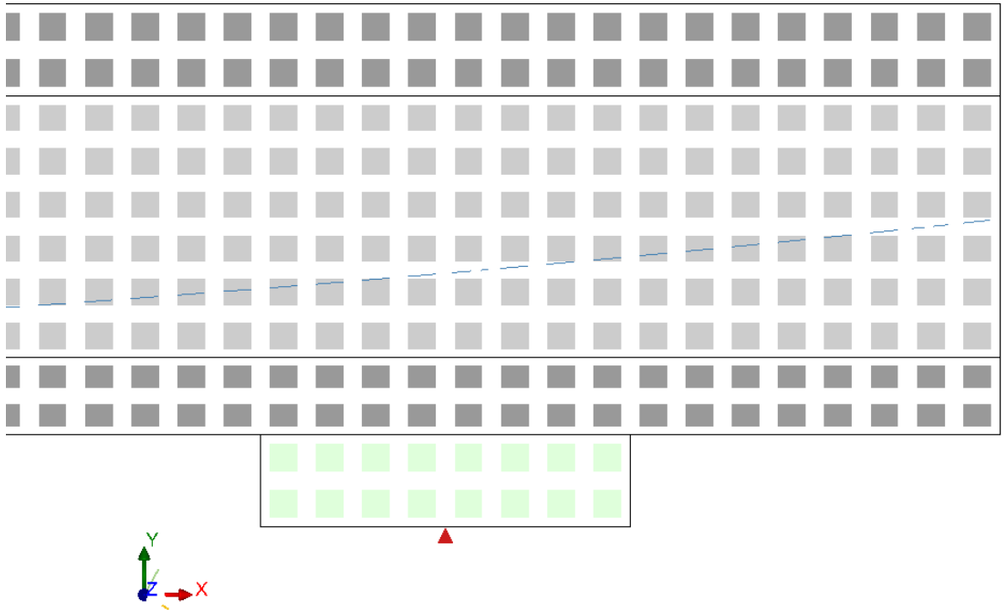


Figure 56: Mesh at right side
(anchor reaction force

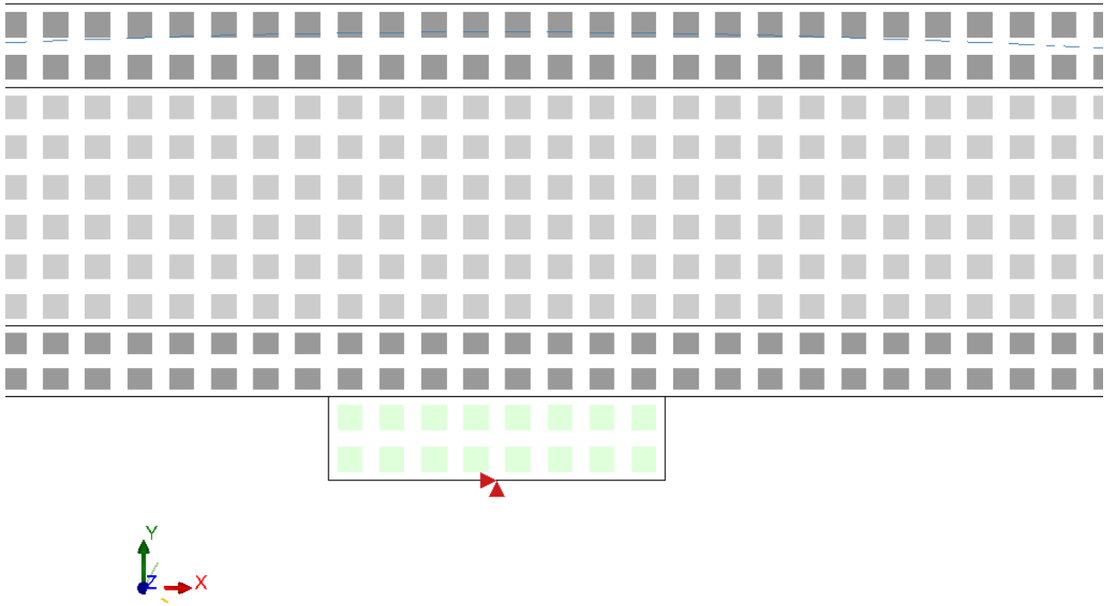


Figure 57: Mesh at mid-span

3 Phased Nonlinear Static Analysis

A phased analysis is required to take into account the stress history of the post-tensioning sequence. For this tutorial, two different phases are required: *i*) jacking of the tendon and *ii*) locking and grouting of the tendon, applying additional local loads. These phases are detailed in Table 2.

Table 2: Sequence of the phased analysis

Phase name	Description	Bond properties	Analysis type	Execution block type	Load combination
Phased - jacking	jacking of tendon	weak	nonlinear analysis	load steps	4 (gravity, post-tensioning loads)
Phased - locked	locked & grouted tendon	stiff	nonlinear analysis	start steps load steps	1 (gravity) 3 (local load)

To set up a *phase* we follow these steps:

1. create a new *Phase* (rename accordingly)
2. open the *Edit properties* dialog box to:
   - select the *Element sets* to be active during the phase
   - select the element material properties (if required)
   - select the *Support sets* to be active during the phase (if required)
3. add the required analysis commands (in this tutorial *Structural nonlinear*):
   - add a *Start step* if new elements are included in the model
     (here, this step is not necessary for the first phase and is skipped)
   - set up the *Load step* (or remove it if not needed in the phase)
   - set up the calculations parameters (e.g., solver, convergence criteria, superposition, etc.)

3.1 Commands

We start by adding a new analysis and renaming it as *Bondslip phased* .

3.1.1 Phase 1: Activate Post-Tensioning Loads

We add the first phase where we apply the jacking of the tendon. The tendon is not locked at the anchors and is not grouted. Therefore, we keep the already assigned weak bond material properties [Table 1].

DIANAIE

Main menu → Analysis → Add analysis

Analysis browser → Analysis1 → Rename → Bondslip phased [Fig. 58]

Analysis browser → Bondslip phased → Add command → Phased [Fig. 59]

Analysis browser → Bondslip phased → Phased → Rename → Phased - post-tensioning

Main menu → Analysis → Edit phases [Fig. 60] [Fig. 61]

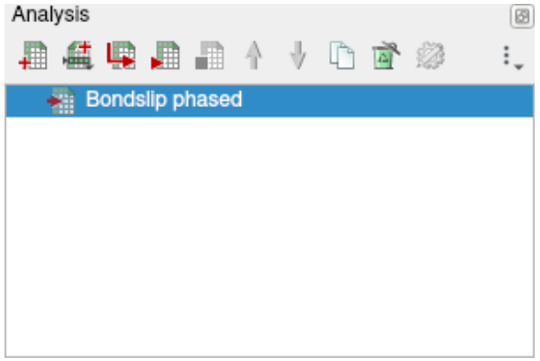


Figure 58: Analysis browser

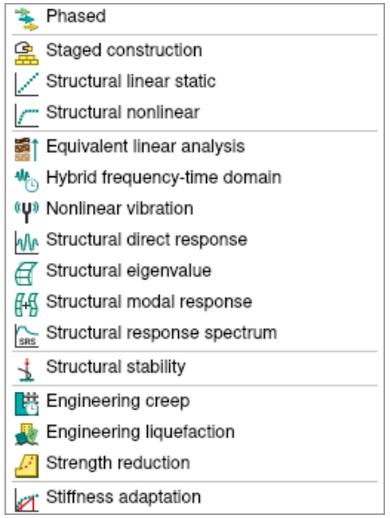


Figure 59: Add analysis command

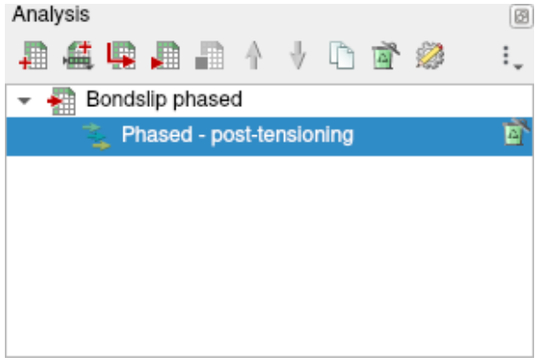


Figure 60: Analysis browser - edit phase properties

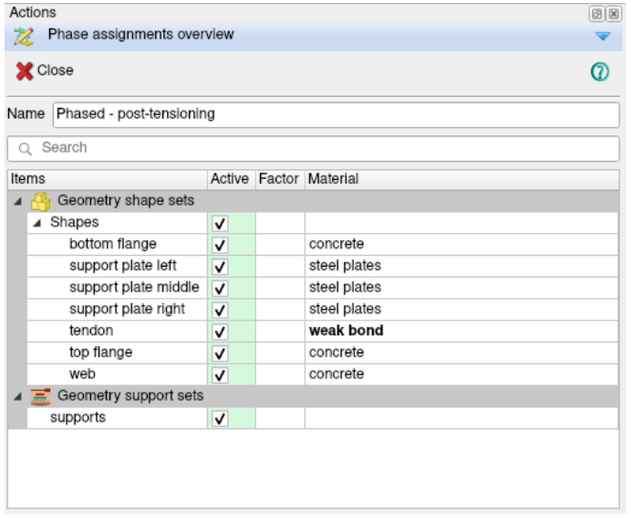


Figure 61: Edit phased properties

3.1.2 Phase 1: Structural Nonlinear Analysis

We add a *Structural nonlinear* analysis command to the first phase. The gravity and the post-tensioning loads are acting on the system (*load combination 4*). The loading is carried out in 1 step.

DIANAIE

Analysis browser → Bondslip phased → Add command → Structural nonlinear [Fig. 62]

Analysis browser → Bondslip phased → Structural nonlinear → new execute block → Load steps → Edit properties [Fig. 63] [Fig. 64]

Analysis browser → Bondslip phased → Structural nonlinear → new execute block → Equilibrium iteration → Edit properties [Fig. 63] [Fig. 65]

Analysis browser → Bondslip phased → Structural nonlinear → new execute block → Output → Edit properties [Fig. 63] [Fig. 66]

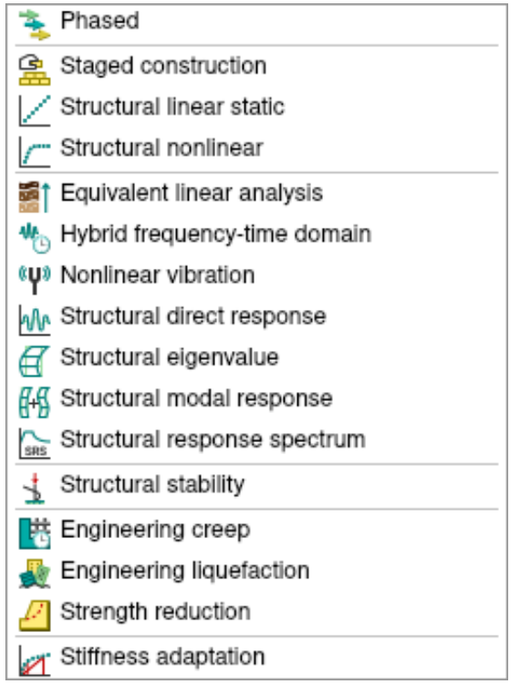


Figure 62: Add analysis command

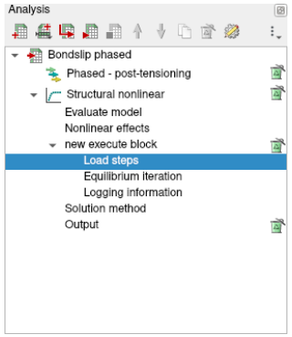


Figure 63: Analysis browser

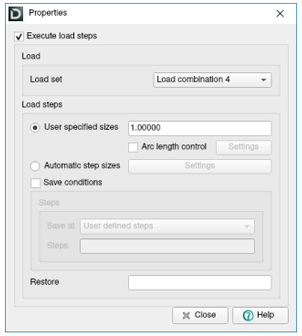


Figure 64: Edit load steps

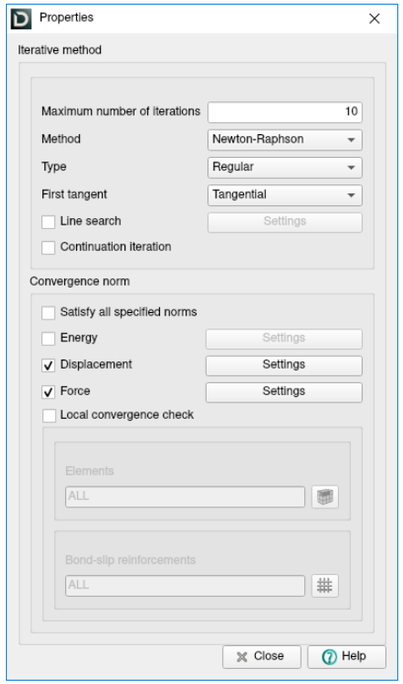


Figure 65: Edit equilibrium iteration (default)

Figure 66: Edit output (default)

3.1.3 Phase 2: Activate Locking of Tendon

We add a second phase for the analysis of the locked and grouted of the tendon. Now, the tendon is assigned with the stiff bond material properties [Table 1].

DIANAIE

Analysis browser → Bondslip phased → Add command → Phased [Fig. 67]
Analysis browser → Bondslip phased → Phased → Rename → Phased - locked [Fig. 58]
Main menu → Analysis → Edit phases [Fig. 68] [Fig. 69]

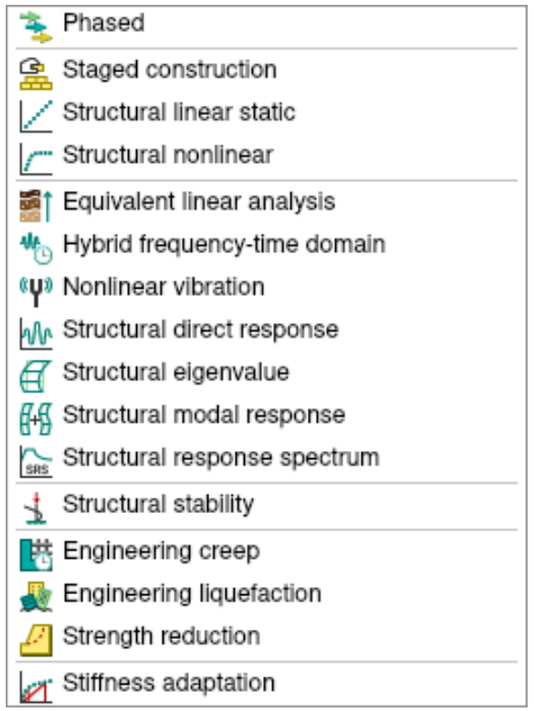


Figure 67: Add analysis command

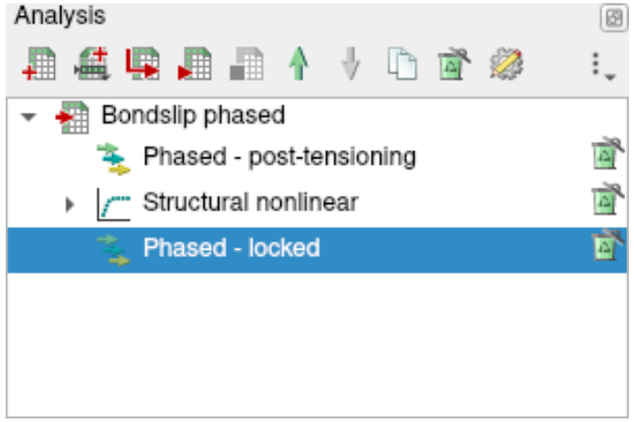


Figure 68: Analysis browser - edit phased properties

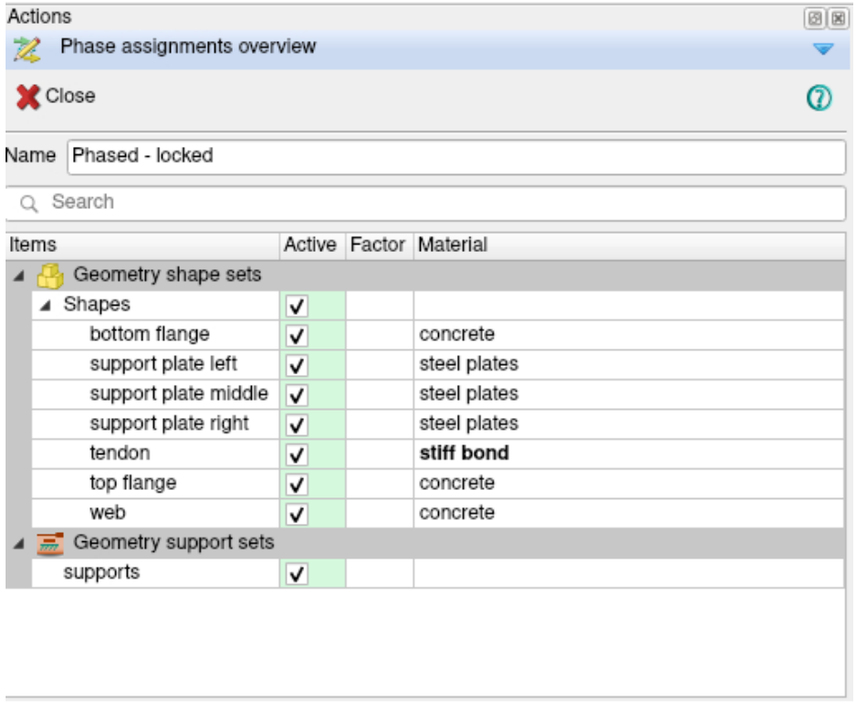


Figure 69: Edit phased properties

3.1.4 Phase 2: Structural Nonlinear Analysis - Start Steps

We add a *Structural nonlinear* analysis command to the second phase. We must first initialize the stresses due to the already acting - from the previous phase - gravity load. Therefore, we change the existing *Load steps* execute block into a *Start steps* execute block, where we apply the gravity load (*load combination 1*) [Fig. 73] in one load step.
The post-tensioning loads do not act in this phase (included in *load combination 4*). Therefore, the *Use load of previous phase* option is de-activated in the *Start steps* properties dialog [Fig. 73].

DIANAIE

Analysis browser → Bondslip phased → Add command → Structural nonlinear [Fig. 70]
Analysis browser → Bondslip phased → Structural nonlinear 1 → new execute block [Fig. 71]
Property Panel → Description → Step type → Value → Start steps [Fig. 72]
Analysis browser → Bondslip phased → Structural nonlinear 1 → new execute block → Start steps → Edit properties [Fig. 72] [Fig. 73]

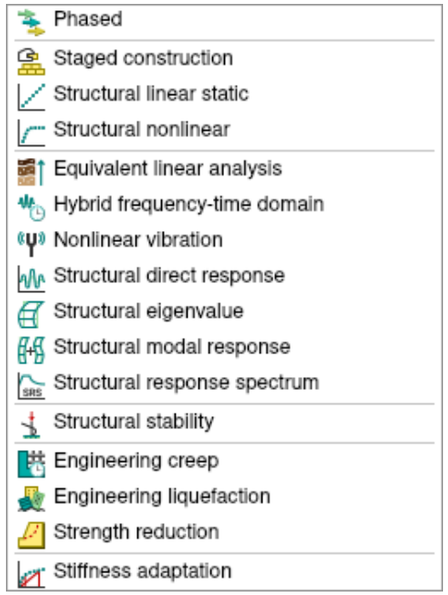


Figure 70: Add analysis command

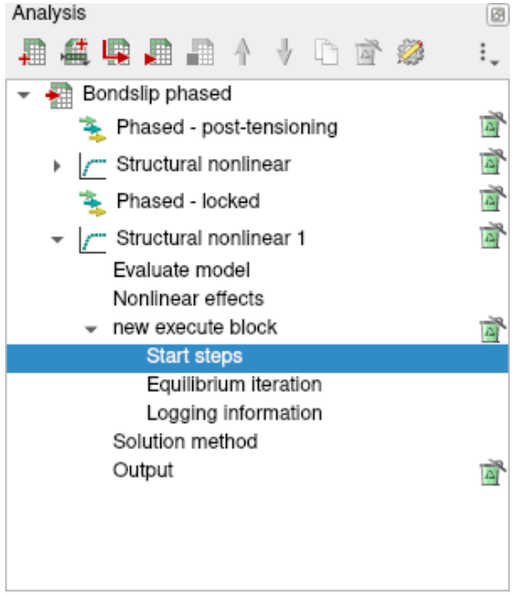


Figure 71: Analysis brower - add start steps

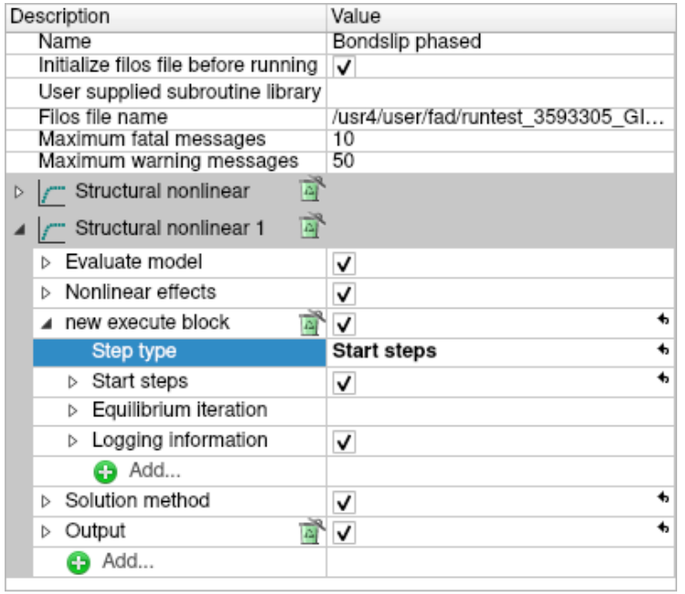


Figure 72: Analysis properties

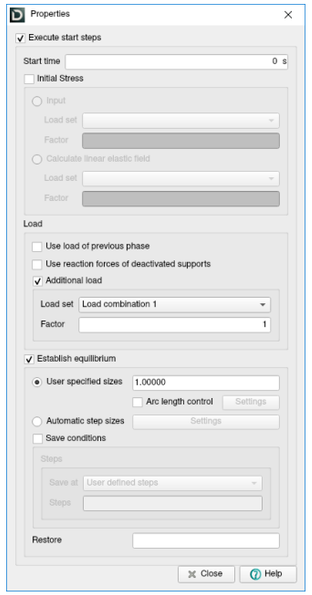


Figure 73: Edit start steps

Note that the defaults are selected for the *Equilibrium iteration* of the current execute block similarly to Figure 65.

3.1.5 Phase 2 - Structural Nonlinear Analysis - Load Steps

We apply the local load (*load combination 3*) by adding a new *Load steps* execute block to the nonlinear analysis. The load is applied in one step. Finally, we run the *Bondslip phased* analysis.

DIANAIE

Analysis browser → Bondslip phased → Structural nonlinear 1 → Add... → Execute steps - Load steps [Fig. 74] [Fig. 75]

Analysis browser → Bondslip phased → Structural nonlinear 1 → new execute block 2 → Load steps → Edit properties [Fig. 75] [Fig. 76]

Analysis browser → Bondslip phased → Structural nonlinear 1 → new execute block 2 → Equilibrium iteration → Edit properties [Fig. 75] [Fig. 77]

Main menu → Analysis → Run selected analysis

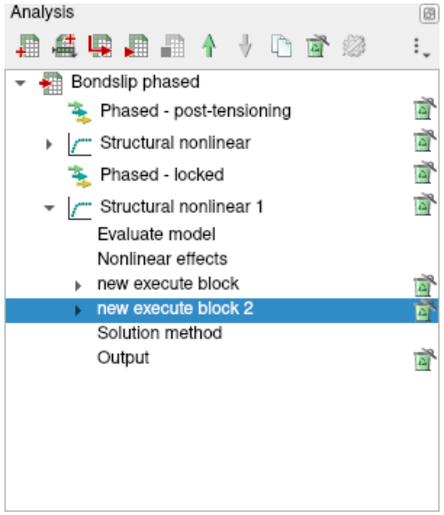


Figure 74: Analysis browser - load steps

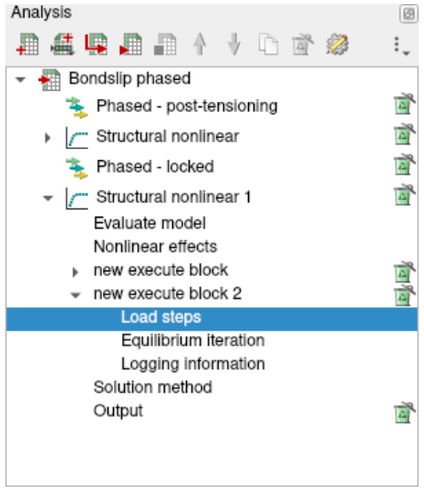


Figure 75: Analysis browser

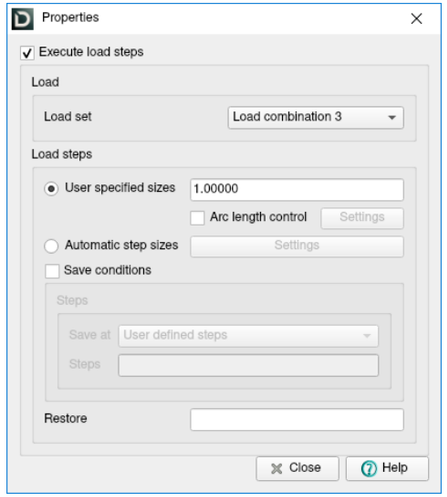


Figure 76: Edit load steps

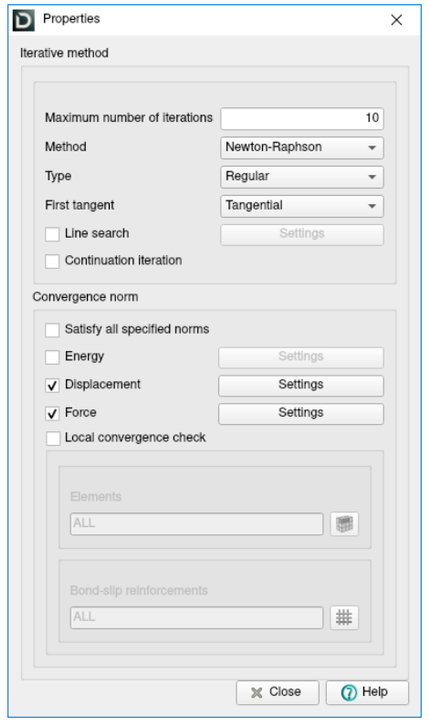


Figure 77: Edit equilibrium iterations

Note that the default is selected for the *Output* of *Structural nonlinear analysis 1* similarly to Figure 66.

3.2 Results

3.2.1 Vertical Displacements

First, we examine the vertical displacements TDtY of the beam during the phases of post-tensioning.

DianaIE

Results browser → Bondslip phased → Output → Nodal results → Total displacements → TDtY → Show line diagram
Results browser → Case → Phased - post-tensioning, Load-step 1, Load-factor 1.0000 [Fig. 78]
Results browser → Case → Phased - locked, Start-step 1, Load-factor 1.0000 [Fig. 79]
Results browser → Case → Phased - locked, Load-step 2, Load-factor 1.0000 [Fig. 80]

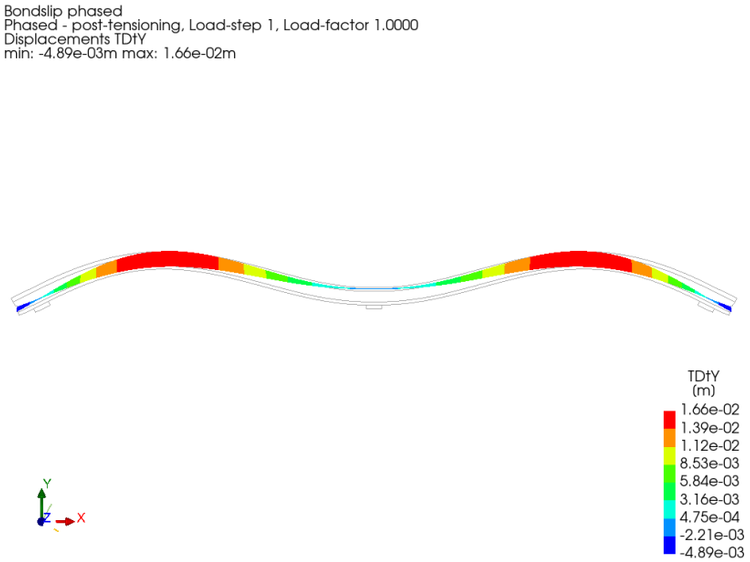


Figure 78: Displacements TDtY - Phase 1, Step 1

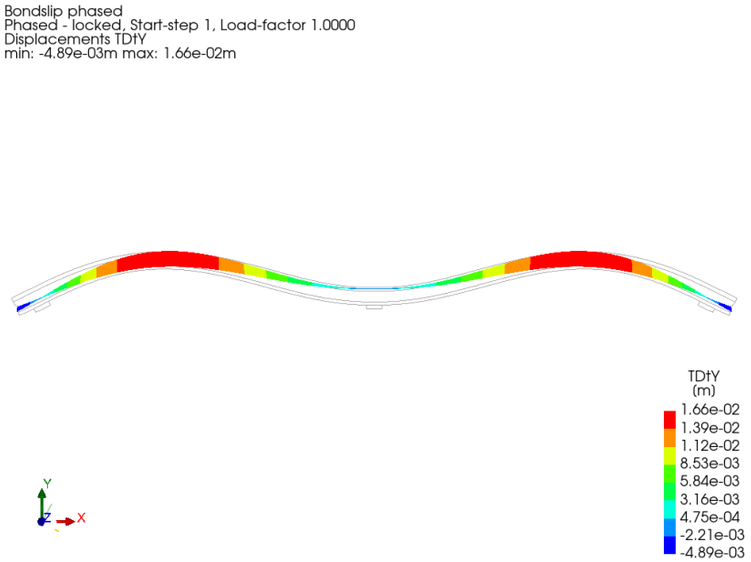


Figure 79: Displacements TDtY - Phase 2, Step 1

Figure 80: Displacements TDtY - Phase 2, Step 2

3.2.2 Reinforcement Stresses

Next, we examine the reinforcement stresses SXX during the phases of post-tensioning.

DIANAIE

Results browser → Bondslip phased → Output → Reinforcements results → Reinforcement Cauchy Total Stresses → SXX → Show line diagram [Fig. 81]
Results browser → Case → Phased - post-tensioning, Load-step 1, Load-factor 1.0000 [Fig. 82]
< *Repeat for the next two steps* > [Fig. 82] [Fig. 83]

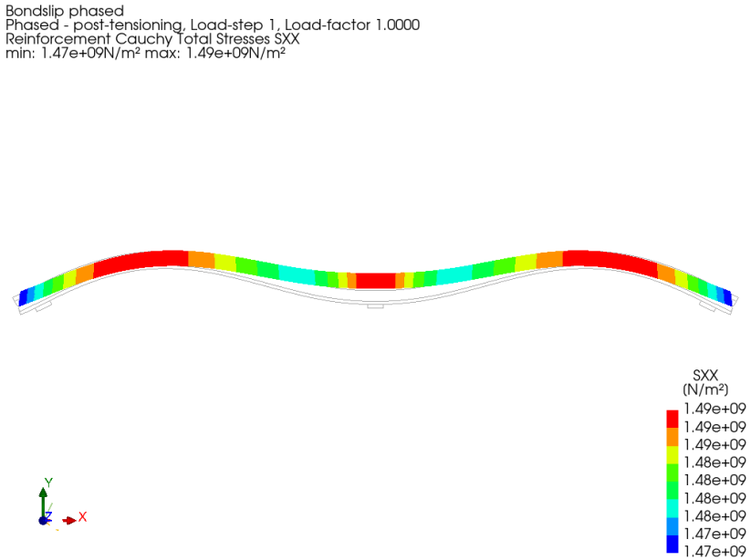


Figure 81: Reinforcement stresses SXX - Phase 1, Step 1

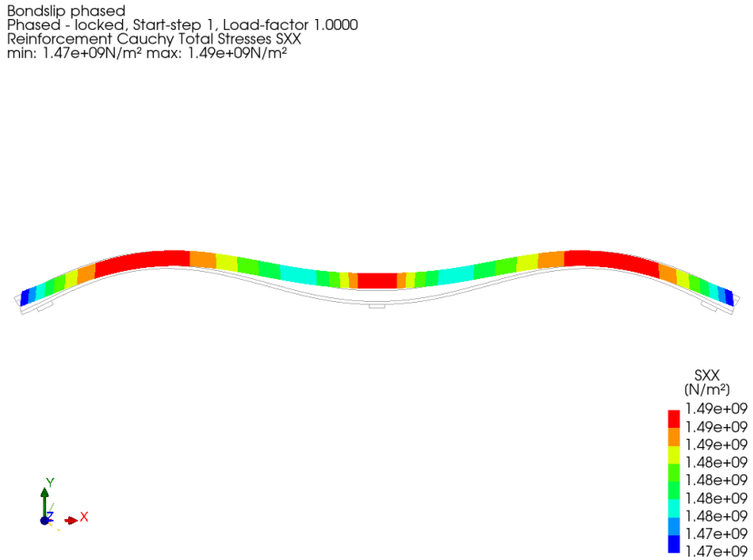


Figure 82: Reinforcement stresses SXX - Phase 2, Step 1

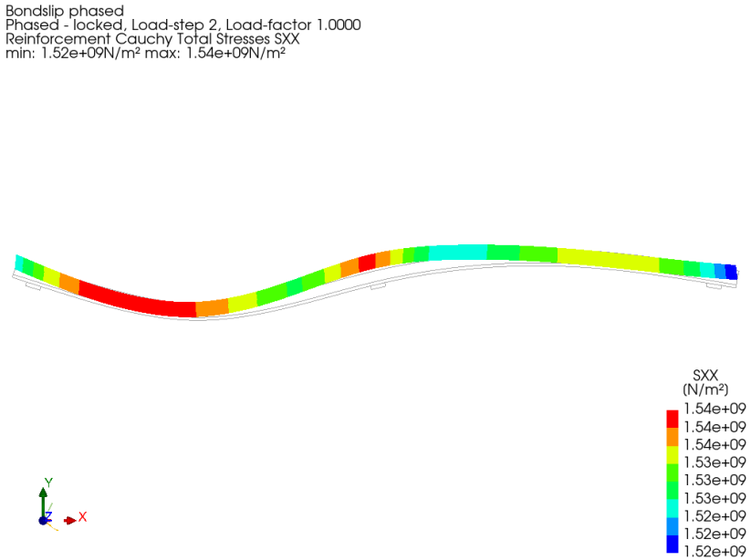


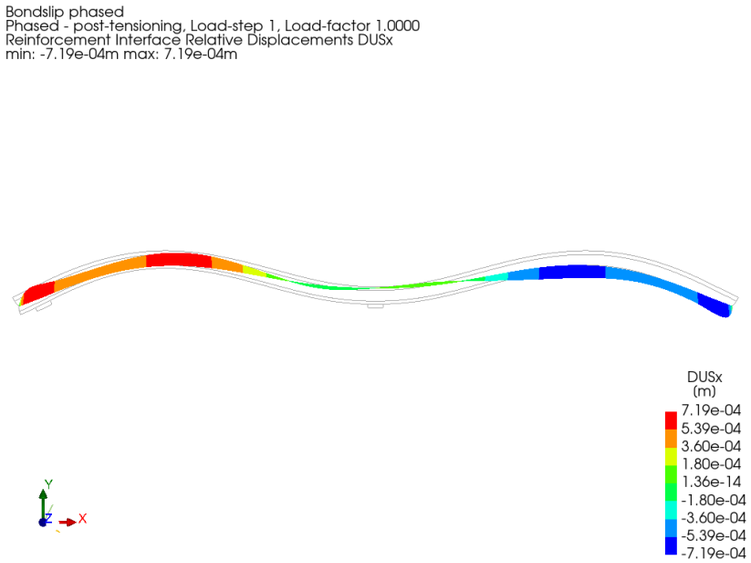
Figure 83: Reinforcement stress SXX - Phase 2, Step 2

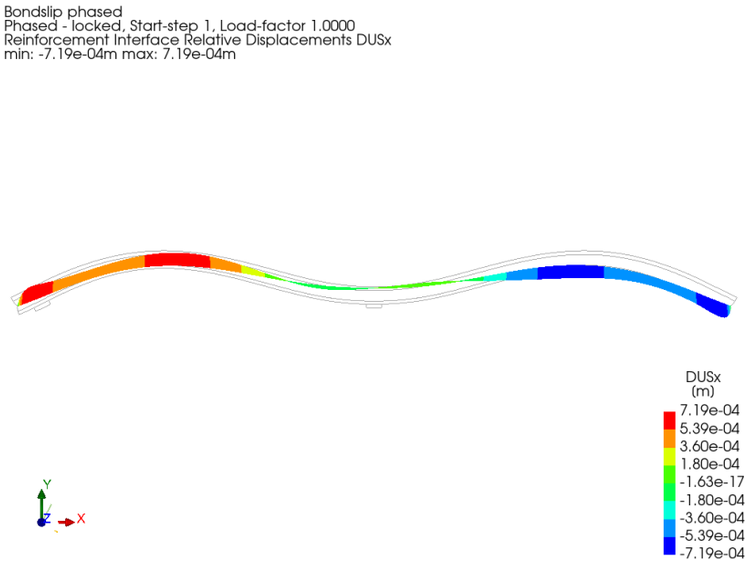
3.2.3 Shear Slip along Reinforcement

We examine the relative displacement of the bondslip reinforcement DUSx during the phases of post-tensioning.

DIANAIE

Results browser → Bondslip phased → Output → Reinforcements results → Reinforcement Interface Relative Displacement → DUSx → Show line diagram [Fig. 84]
Results browser → Case → Phased - post-tensioning, Load-step 1, Load-factor 1.0000 [Fig. 85]
< *Repeat for the next two steps* > [Fig. 85] [Fig. 86]





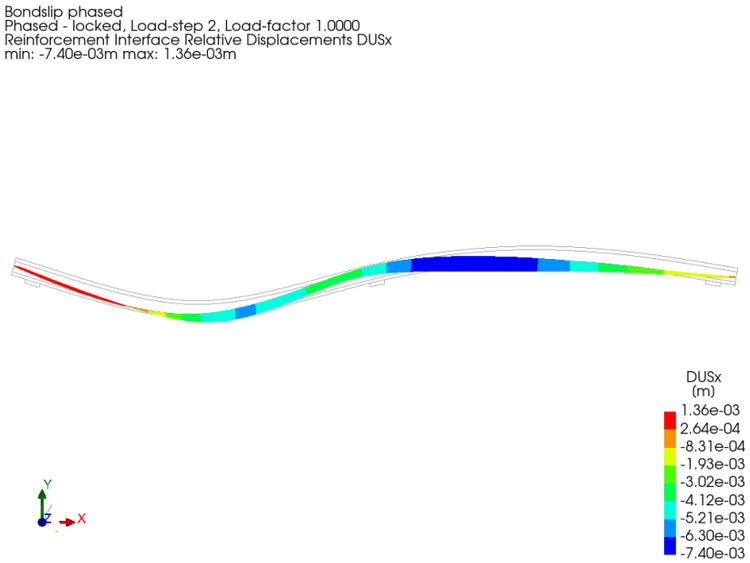


Figure 84: Shear slip along reinforcement DUSx - Phase 1, Figure 85: Shear slip along reinforcement DUSx - Phase 2, Step 1, Figure 86: Shear slip along reinforcement DUSx - Phase 2, Step 2

3.2.4 Shear Stress in the Bond-Slip Interface

We examine the shear stress in the tendon concrete interface bond STSx during the phases of post-tensioning. In phase 1 the shear stress in the bond-slip interface is very low due to the low shear stiffness. In phase 2 step 1, the shear stresses are still very low due to the option *reset state parameters on material change* [Fig. 38].

DIANAIE

Results browser → Bondslip phased → Output → Reinforcements results → Reinforcement Interface Tractions → STSx → Show line diagram [Fig. 87]
Results browser → Case → Phased - post-tensioning, Load-step 1, Load-factor 1.0000 [Fig. 88]

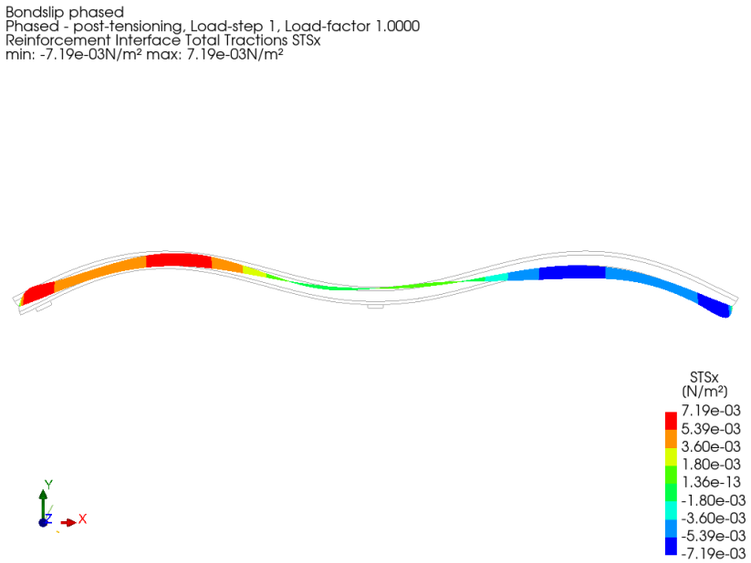


Figure 87: Shear stress in tendon-concrete interface STSx - Phase 1, Step 1

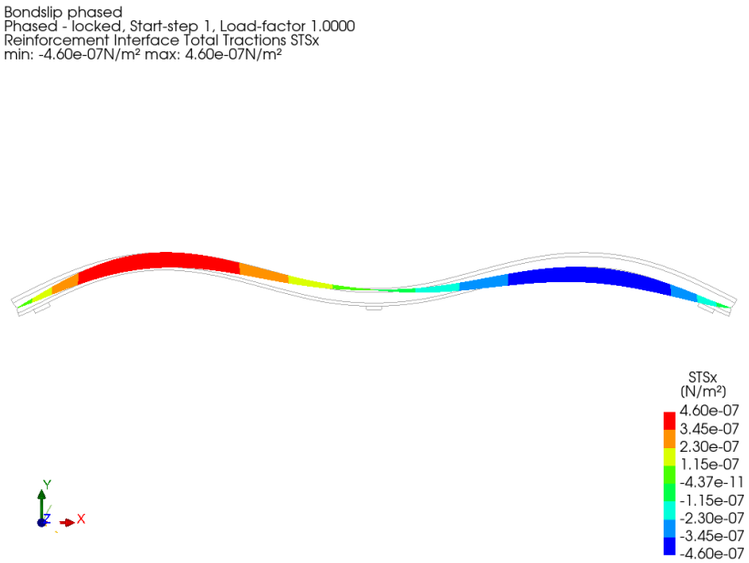


Figure 88: Shear stress in tendon-concrete interface STSx- Phase 2, Step 1

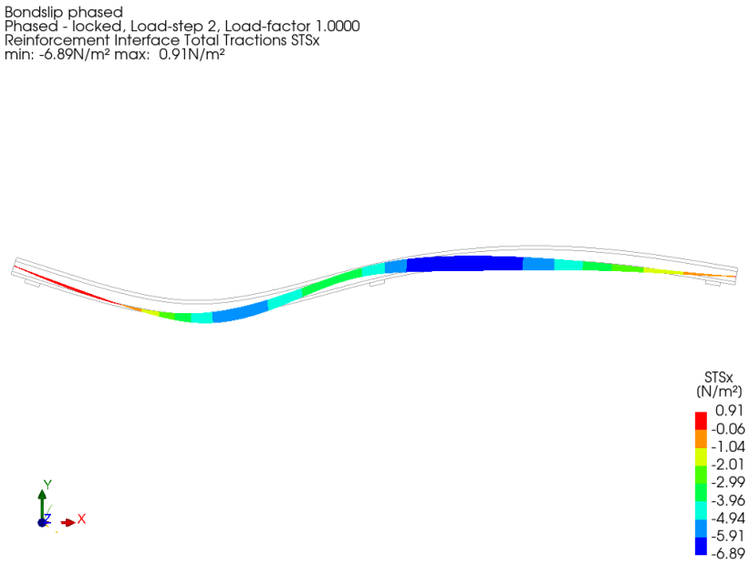


Figure 89: Shear stress in tendon-concrete interface STSx - Phase 2, Step 2

Appendix A Additional Information

Folder: Tutorials/BondslipPostTensionReinfo2D

Number of elements ≈ 4454

Keywords:
ANALYS: nonlin phase physic.
CONSTR: suppor.
ELEMEN: bar bondsl enhanc interf l4tru pstres q10if q12if q8mem reinfo struct truss.
LOAD: force node prestr reinfo weight.
MATERI: concre crack elasti isotro mc2010 slip.
OPTION: direct newton regula units.
POST: binary ndiana.
PRE: dianai.
RESULT: cauchy displa extern force green reacti strain stress total tracti.

Laan van Waalhaven 462
2497 GR, The Hague
The Netherlands

DIANAFEA.COM INFO@DIANAFEA.COM +31(0)88 34 262 00

DIANA