Bond-Slip of Post-Tensioned Reinforcement in Two-Dimensional Elements
# Outline

## 1 Description
- 1.1 Modeling Approach ................................................... 4
- 1.2 Material Properties .................................................. 5

## 2 Finite Element Model
- 2.1 Geometry .................................................................. 8
  - 2.1.1 Beam ..................................................................... 8
  - 2.1.2 Reinforcement ....................................................... 9
  - 2.1.3 Support Plates ...................................................... 10
  - 2.1.4 Anchor Plates ....................................................... 12
- 2.2 Properties .................................................................. 13
  - 2.2.1 Concrete Beam ..................................................... 13
  - 2.2.2 Tendon ................................................................. 15
  - 2.2.3 Support Plates ..................................................... 18
  - 2.2.4 Anchor Plates ....................................................... 19
- 2.3 Boundary Conditions ................................................... 20
- 2.4 Loads ..................................................................... 21
  - 2.4.1 Gravity Load .......................................................... 21
  - 2.4.2 Post-Tensioning Load ............................................. 22
  - 2.4.3 Reaction Force at Anchors ...................................... 23
  - 2.4.4 Local Load ........................................................... 24
  - 2.4.5 Load Combinations ................................................. 25
- 2.5 Tyings .................................................................... 26
- 2.6 Mesh ..................................................................... 27

## 3 Phased Nonlinear Static Analysis
- 3.1 Commands ................................................................. 30
  - 3.1.1 Phase 1: Activate Post-Tensioning Loads ...................... 30
  - 3.1.2 Phase 1: Structural Nonlinear Analysis ....................... 31
  - 3.1.3 Phase 2: Activate Locking of Tendon .......................... 32
  - 3.1.4 Phase 2: Structural Nonlinear Analysis - Start Steps .... 33
  - 3.1.5 Phase 2: Structural Nonlinear Analysis - Load Steps .... 34
- 3.2 Results .................................................................. 35
  - 3.2.1 Vertical Displacements ............................................ 35
  - 3.2.2 Reinforcement Stresses ............................................ 36
  - 3.2.3 Shear Slip along Reinforcement .................................. 37
  - 3.2.4 Shear Stress in the Bond-Slip Interface ...................... 38

## Appendix A Additional Information ............................................. 39
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 reaction forces at the anchors
- post-tensioning load on embedded (no-bond) reinforcement

The first solution is applied in this tutorial, since we attempt to realistically model the described phases of post-tensioning. Therefore, we have to take into account the slipping and friction losses of the tendon in the duct and the reaction forces due to the jacking forces.

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 embedded 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 three 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 by two nodal forces at both ends, simultaneously. Weak bonding is defined between tendon and concrete since the tendon is still not grouted. Counterbalance reaction forces, due to jacking, are applied over the anchor locations. The anchor plates are not there yet.
  Phase 2:
  - the tendon is now grouted. The weak bond interface properties are changed into stiff elastic properties. The jacking loads are removed. Tyings are applied to lock the tendon at the anchor locations. The anchor plates are added.
  - a local vertical load is applied at the middle of the left span.

![Figure 2: Cubic function by Doerr](https://dianafea.com)
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.

<table>
<thead>
<tr>
<th>Concrete</th>
<th>Model code</th>
<th>fib Model Code 2010</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grade</td>
<td>C55</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Reinforcement steel (elastic)</th>
<th>Young's modulus $E_x$</th>
<th>2.1e+11 N/m$^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.3</td>
<td></td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>7.85E+3 kg/m$^3$</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Bond-slip reinforcement interface</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weak bond (empty duct - Phase 1)</td>
</tr>
<tr>
<td>Normal stiffness modulus $D_{11}$</td>
</tr>
<tr>
<td>Shear stiffness modulus $D_{22}$</td>
</tr>
<tr>
<td>Failure model cubic function by Doerr</td>
</tr>
<tr>
<td>Parameter c</td>
</tr>
<tr>
<td>Shear slip at start plateau</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Stiff bond (grouted - Phase 2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal stiffness modulus $D_{11}$</td>
</tr>
<tr>
<td>Shear stiffness modulus $D_{22}$</td>
</tr>
<tr>
<td>Failure model cubic function by Doerr</td>
</tr>
<tr>
<td>Parameter c</td>
</tr>
<tr>
<td>Shear slip at start plateau</td>
</tr>
</tbody>
</table>
2 Finite Element Model

For the modeling session we start a new project for a two dimensional structural analysis. Quadrilateral mesh with linear interpolation is defined. The *Model size* is set to 1 km (-500 to 500m) to include the entire model ($X=\pm 60$ m).

![New project dialog](https://dianafea.com)
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.

**Geometry browser — Reference system — Units**  [Fig. 4]

**Property Panel**  [Fig. 5]

---

**Figure 4:** Geometry browser - units

**Figure 5:** Properties panel - units
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 $60 \times 0.25 \text{ m}$, $60 \times 0.85 \text{ m}$ and $60 \times 0.30 \text{ m}$, respectively.

---

**Figure 6**: Add polygon sheet - bottom flange

**Figure 7**: Add polygon sheet - web

**Figure 8**: Add polygon sheet - top flange

**Figure 9**: Geometry - bottom flange, web and top flange
2.1.2 Reinforcement

We draw the curved profile of the tendon.

Main menu ➔ Geometry ➔ Create ➔ Add curve [Fig. 10] [Fig. 11]

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.
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.

Main menu → Geometry → Create → Add polygon sheet
Main menu → Geometry → Create → Add point
Main menu → Geometry → Modify → Shape projection

Note that we use different colors for different shapes

Figure 12: Add polygon sheet: left support
Figure 13: Add vertex
Figure 14: Left support / Add & Project vertex
Figure 15: Project vertex on support edge
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.

---

**Figure 16**: Geometry: beam, tendon and support plates
**Figure 17**: Mirror shape - left support
**Figure 18**: Array copy - left support
**Figure 19**: Middle support plate
**Figure 20**: Right support plate
2.1.4 Anchor Plates

Finally, we create the geometry of the anchor plates. First we create a rectangular sheet for the left anchor plate. We define points at the end of the tendon so that we can use it for tying the tendon to the anchor plate in phase 2. Then we mirror the anchor plate to get the right anchor plate. We imprint the anchor plates on the web so that we can add the counterbalance reaction forces, due to jacking, on the web at the location of the anchor 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]. We define the element geometry with thickness equal to the bottom flange width (0.8 m).
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. 27 to 28]). We may set the local axis (default X axis).

**Main menu**  ➔  **Geometry**  ➔  **Assign**  ➔  **Shape Properties**  ➔  [Fig. 30]  [Fig. 33]
**Shape Properties**  ➔  ➔  **Geometry**  ➔  **Add new geometry**  ➔  [Fig. 31]  [Fig. 32]
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. 34]

Reinforcement properties ➔ Material ➔ Add material [Fig. 35] ➔ Edit material [Fig. 36] [Fig. 37]
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. 38] for the interface elements; thus, letting DIANA automatically calculate it by using the diameter defined for the bar elements.

Figure 38: 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. 35 to 37]. 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.
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).
2.2.4 Anchor Plates

Finally, we assign properties to the anchor plates. For these anchor plates we use the already defined steel plate material model. For the geometrical properties we use the thickness of the web.

Main menu → Geometry → Assign → Shape Properties [Fig. 46]

Figure 46: Property assignments - Anchor 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.

Main menu ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 47] [Fig. 48] [Fig. 49]

Figure 47: Attach support - vertical support

Figure 48: Attach support - horizontal support

Figure 49: Supports - horizontal and vertical
2.4 Loads

2.4.1 Gravity Load

We apply the dead weight as a global load.

Main menu ➔ Geometry ➔ Assign ➔ Add global loads ➔ [Fig. 50]
2.4.2 Post-Tensioning Load

The post-tensioning is applied at both ends of the tendon, simultaneously. The jacking force at each end point lies in the respective outwards tangential direction [Fig. 57]. The two jacking forces are assigned to the load case "post-tensioning loads". We hide the shapes of Anchor Plate Left and Anchor Plate Right to easier select the both end points of the tendon.

DianaFE
2.4.3 Reaction Force at Anchors

For each jacking force counterbalance reaction forces develop over the anchor surface [Fig. 62]. The two reaction loads are also assigned to the load case "post-tensioning loads". Note that, by not including these reaction loads, the post-tensioning losses due to shortening of the beam are not taken into account.
2.4.4 Local Load

A concentrated force is applied at the left middle span. To do this, we first add and project a vertex [Fig. 64 to 65] on which we attach the point load [Fig. 66] 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.

---

**Figure 63**: Local (point) load

**Figure 64**: Add vertex (left span)

**Figure 65**: Projection of vertex

**Figure 66**: Attach point load
2.4.5 Load Combinations

The defined load cases of gravity and post-tensioning loads act simultaneously during the 1st 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. 68].

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.

---

**Main menu ➔ Geometry ➔ Loads ➔ Open geometry load combinations table** [Fig. 67]

**Geometry Load Combinations ➔ Add geometry load combination ➔ Geometry Load combination 4** [Fig. 68]

---

Figure 67: Geometry load combinations

Figure 68: Geometry load combinations table
2.5 Tyings

After jacking, the tendon is locked at the anchor locations. Thereafter, the locked part of the reinforcement follows the same displacements with the anchor. To model this behavior, we apply tyings between the end points of the tendon (Master) and the respective anchor point of the anchor plate (Slave). The tyings act in the X and Y translational directions.

Figure 69: Attach tying - left anchor
Figure 70: Master point (left anchor)
Figure 71: Slave point (left anchor plate)
Figure 72: Attach tying - right anchor
Figure 73: Master point (right anchor)
Figure 74: Slave point (right anchor plate)
2.6 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.

**Figure 75: Set mesh properties**

**Figure 76: Mesh at left side**

(anchor reaction force)
Note that the *Mesher type* and the *Mid-side node location* have already been defined in the *Project settings* dialog, [Fig. 3]. It is not necessary to re-define them in *Set mesh properties*, in [Fig. 75].
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

<table>
<thead>
<tr>
<th>Phase name</th>
<th>Description</th>
<th>Bond properties</th>
<th>Analysis type</th>
<th>Execution block type</th>
<th>Load combination</th>
</tr>
</thead>
<tbody>
<tr>
<td>Phased - jacking</td>
<td>jacking of tendon</td>
<td>weak</td>
<td>nonlinear analysis</td>
<td>load steps</td>
<td>4 (gravity, post-tensioning loads)</td>
</tr>
<tr>
<td>Phased - locked</td>
<td>locked &amp; grouted tendon</td>
<td>stiff</td>
<td>nonlinear analysis</td>
<td>start steps</td>
<td>1 (gravity)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>load steps</td>
<td>3 (local load)</td>
</tr>
</tbody>
</table>

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 and Tying 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]. We do not activate the elements of the anchor plates.

Main menu → Analysis → Add analysis [Fig. 79]
Analysis browser → Analysis1 → Rename → Bondslip phased [Fig. 79]
Analysis browser → Bondslip phased → Add command → Phased [Fig. 80]
Analysis browser → Bondslip phased → Phased → Rename → Phased - post-tensioning
Analysis browser → Bondslip phased → Phased - post-tensioning → Edit phases [Fig. 81] [Fig. 82]
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.

Figure 83: Add analysis command

Figure 84: Analysis browser

Figure 85: Edit load steps

Figure 86: Edit equilibrium iteration (default)

Figure 87: 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 tendon. Now, the tendon is assigned with the stiff bond material properties [Table 1]. We also activate the anchor plates.

---

**Analysis browser** ➔ Bondslip phased ➔ Add command ➔ Phased [Fig. 88]
**Analysis browser** ➔ Bondslip phased ➔ Phased ➔ Rename ➔ Phased - locked [Fig. 79]
**Analysis browser** ➔ Bondslip phased ➔ Phased - locked ➔ Edit phases [Fig. 99] [Fig. 100]

**Figure 88:** Add analysis command

**Figure 89:** Analysis browser - edit phased properties

**Figure 90:** 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. 94] 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. 94].

**Note that the defaults are selected for the *Equilibrium iteration* of the current execute block similarly to Figure 86.**
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.

Analysis browser ➔ Bondslip phased ➔ Structural nonlinear 1 ➔ Add... ➔ Execute steps - Load steps ➔ [Fig. 95] [Fig. 96]
Analysis browser ➔ Bondslip phased ➔ Structural nonlinear 1 ➔ new execute block 2 ➔ Load steps ➔ Edit properties ➔ [Fig. 96] [Fig. 97]
Analysis browser ➔ Bondslip phased ➔ Structural nonlinear 1 ➔ new execute block 2 ➔ Equilibrium iteration ➔ Edit properties ➔ [Fig. 96] [Fig. 98]

Main menu ➔ Analysis ➔ Run selected analysis

Note that the default is selected for the Output of Structural nonlinear analysis 1 similarly to Figure 87.
3.2 Results

3.2.1 Vertical Displacements

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

Figure 99: Displacements TDtY - Phase 1, Step 1
Figure 100: Displacements TDtY - Phase 2, Step 1
Figure 101: Displacements TDtY - Phase 2, Step 2
3.2.2 Reinforcement Stresses

Next, we examine the reinforcement stresses $S_{XX}$ during the phases of post-tensioning.

---

**Results browser** → Bondsip phased → Output → Reinforcements results → Reinforcement Cauchy Total Stresses → $S_{XX}$ → Show line diagram

**Results browser** → Case → Phased - post-tensioning, Load-step 1, Load-factor 1.0000

< Repeat for the next two steps >

---

Figure 102: Reinforcement stresses $S_{XX}$ - Phase 1, Step 1

Figure 103: Reinforcement stresses $S_{XX}$ - Phase 2, Step 1

Figure 104: Reinforcement stress $S_{XX}$ - Phase 2, Step 2

Bondslip Reinforcement in 2D | https://dianafea.com

36/40
3.2.3 Shear Slip along Reinforcement

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

Results browser ➔ Bondslip phased ➔ Output ➔ Reinforcements results ➔ Reinforcement Interface Relative Displacement ➔ DUSx ➔ Show line diagram

Results browser ➔ Case ➔ Phased - post-tensioning, Load-step 1, Load-factor 1.0000

< Repeat for the next two steps >

Figure 105: Shear slip along reinforcement DUSx - Phase 1, Step 1

Figure 106: Shear slip along reinforcement DUSx - Phase 2, Step 1

Figure 107: 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. 41].

Figure 108: Shear stress in tendon-concrete interface STSx - Phase 1, Step 1

Figure 109: Shear stress in tendon-concrete interface STSx - Phase 2, Step 1

Figure 110: Shear stress in tendon-concrete interface STSx - Phase 2, Step 2
Appendix A  Additional Information

Folder: Tutorials/BondslipReinfo2D

Number of elements ≈ 4454

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