Embedded Reinforcements
Gerd-Jan Schreppers, January 2011

Abstract: This paper explains the concept and application of embedded reinforcements in DIANA. Basic assumptions and definitions, the pre-processing for bars and grids in combination with different element-families will be discussed, as well pre-stress loadings, post-tensioning functions and specific material models for reinforcements.

Introduction
In DIANA a reinforced concrete structure can be modelled by plain concrete elements and steel-reinforcement bars. When the steel reinforcement is composed of a series of bars which are located at a fixed intermediate distance from each-other, this can be modelled as a reinforcement grid. Bars in a grid may be located all in the same direction or being oriented in two orthogonal directions.

The elements and the reinforcements can be defined independent from each other, each with reference to their own geometry definition.

To the concrete elements different nonlinear material properties will be assigned than to the steel reinforcements. The concrete material model shall account for cracking failure under tensile stresses and crushing failure at compressive and shear stresses. The steel reinforcements can be modelled with von-Mises-type elasto-plastic material models with user-defined hardening.

Reinforcements are usually fully embedded in the concrete elements and by default displacements and strains of reinforcements and elements are fully coupled. Alternatively, Bond-slip Reinforcements can be chosen, which are explained in a separate Technical Paper.

Basic Assumptions and Definitions
Embedded reinforcements add stiffness to the finite element model. Their main characteristics are:

- Reinforcements are embedded in structural elements, the so-called mother elements.
- Reinforcements do not have degrees of freedom of their own.
- By default, reinforcement strains are computed from the displacement field of the mother elements. This implies perfect bond between the reinforcement and the surrounding material. However, with the NOBOND input option you can specify that the reinforcement is not bonded to the embedding elements.

The technique of embedding allows the geometries of the mesh. This permits the user to generate the finite element mesh without having to anticipate on the location of reinforcements.

A reinforcement is defined by the following information:

- location in the model
- material properties
- physical properties (cross-section)
- integration point scheme’s

Special loadings can be applied to reinforcements. DIANA has 2 types of embedded reinforcements:
- bars
- grids

Reinforcement Bars
Reinforcement bars may be embedded in various families of elements: beams, plane stress, curved shell and solid. In finite element models with these elements, bar reinforcements have the shape of a line. Bars may also be embedded in plane strain and axisymmetric elements where they have the shape of a point. The total length of the bar is considered to be divided in several particles. By definition, a particle must be completely inside a structural element.

![Particles, location- and integration-points and stress in a bar.](image)

The so-called location points define the position of the particles in the finite element model. Some location points are the intersections of the bar with the element boundaries. Other location points are in-between these intersections, these points define the curvature of the bar.

Usually, the location points are determined automatically by DIANA from input of larger sections; this process is called pre-processing of reinforcement location. In some cases it may be useful to specify the location points explicitly, which we call element-by-element input.
DIANA performs numerical integration of each particle of a reinforcement bar. In each integration point DIANA determines a vector tangential to the bar axis. The variables for a bar reinforcement are the strains $\varepsilon_{xx}$ and the stresses $\sigma_{xx}$ oriented direction of the vector.

**Reinforcement Grids**

Plane shaped reinforcement grids may be embedded in various families of elements: plane stress, curved shell and solid. Depending on the element family and the specified location points, the plane of the grid may be curved or flat. Grids may also be embedded in plane strain or axisymmetric elements where they have the shape of a line. The total area of the grid is considered to be divided in several *particles*. Each particle contributes to the stiffness of the element that embeds it. The definition of a particle depends on the dimensionality of the embedding structural element.

**Grid reinforcement**

Two-dimensional elements may be fully or partly covered by one or more particles of grid reinforcement. Solid elements embed a particle of grid reinforcement completely. The so-called *location points* define the position of the particles in the finite element model.

Usually, the embedding elements (and for solids the location points) are determined automatically by DIANA from input of larger sections; this process is called *preprocessing* of reinforcement location. In some cases it may be useful to specify the elements (and location points) explicitly, which we call *element-by-element* input.

---

**Embedded Reinforcement Pre-processing**

**Bar in beam elements**

Bar reinforcement can be embedded in beam elements of class-II and class-III. Class-I beam elements cannot embed bar reinforcement. DIANA assumes that a beam element embeds a particle of a reinforcement bar, if the bar intersects two different faces of the beam’s envelope. The envelope of a beam element is defined by six faces: two end faces and four lateral faces.

**Bar in plane stress elements**

Bar reinforcement can be embedded in all plane stress elements. Plane stress elements are automatically checked for embedding of bar reinforcements specified with sections. A plane stress element embeds a particle of a bar section if it intersects one or two element edges, but none of them more than once.

**Bar in plane strain elements**

Reinforcement bars may be embedded in all plane strain elements. Due to the nature of plane strain element models, the bars show up in the $XY$ plane as points.
Bar in axisymmetric elements

Reinforcement bars may be embedded in all axisymmetric regular solid ring elements. Due to the nature of axisymmetry the bars are ring-shaped and show up in the $XY$ plane as points.

Bar reinforcement in axisymmetric element

Bar reinforcement in curved shell elements

Bar reinforcement can be embedded in all curved shell elements. Curved shell elements are automatically checked for embedding of bar reinforcements specified with sections. There are two conditions for a bar section to be embedded in a curved shell element:

1. it must intersect one or two element edges, but none of them more than once;
2. the computed location points must be inside the thickness domain of the element.

Bar reinforcement in curved shell element

Bar reinforcement in solid elements

Bar reinforcement can be embedded in all solid elements. To embed bar reinforcement in solid elements, DIANA needs for each solid element the location points of the particle that is embedded in that element.

Bar reinforcement in solid element

Grid in plane stress elements

Grid reinforcement can be embedded in all plane stress elements. A particle of a grid reinforcement may cover the complete area of the embedding plane stress element or only part of the area of the element. This example specifies a grid reinforcement with one section. The nodes of the section are input with node numbers.

Grid reinforcement in plane stress element

The grid is embedded in a mesh of Q8MEM plane stress elements. DIANA automatically determines which elements are covered completely by the grid (the ones marked with a cross).

Grid in plane-strain elements

Reinforcement grids may be embedded in all plane strain elements. Due to the nature of plane strain the grid shows up in the $XY$ plane as a line between the location points.

Grid reinforcement in plane strain element

Plane strain elements are automatically checked for embedding of reinforcement grids specified with sections. In this case DIANA computes the location points that describe the particles of the grid within the plane strain elements. A grid section is embedded in a plane strain element if it intersects one or two element edges, but none of them more than once.

Grids in axisymmetric elements

Reinforcement grids may be embedded in axisymmetric regular solid ring elements. Due to the nature of axisymmetry the grid shows up in the $XY$ plane as a line between the location points.
Grid reinforcement in axi-symmetric element

Axisymmetric solid ring elements are automatically checked for embedding of reinforcement grids specified with sections. In this case DIANA computes the location points that describe the particles of the grid within the axisymmetric elements. A grid section is embedded in an axisymmetric element if it intersects one or two element edges, but none of them more than once.

Grid in curved shell elements

Reinforcement grids can be embedded in all curved shell elements. A particle of a grid reinforcement may cover the complete area of the embedding curved shell element or part of the area.

The location of the grid particle in the element is determined by location points which must be within the thickness domain of the element. In other words: the eccentricity $z$ cannot be greater than half the thickness $t$ ($z < \frac{1}{2}t$) of the element at the node.

Grid reinforcement in solid elements

This example specifies a grid reinforcement with one section.

Triangularization of grid reinforcement in solid elements

The section contour is a quadrilateral with curved edges, specified with eight nodes. The generated grid particles are triangles with curved edges.

Loadings of Embedded Reinforcements

The most commonly used loading of reinforcements is the pre-stress.

In DIANA the pre-stress loading can be applied as a uniform value for the reinforcement or as a variable pre-stress over the reinforcement. In the latter case the pre-stress is defined by the user in the model-points of the reinforcement and interpolated by the program to the integration points of the reinforcements.

Anchor loadings are not explained in separate Technical Document.
Stiffness Contribution

The contribution of the reinforcement stiffness to the stiffness of the respective mother element is automatically calculated. If for the respective element the option **NOBOND** is defined this reinforcement will not contribute to the stiffness matrix. The **NOBOND** feature can be used to simulate the pre-tensioning of pre-stress-cables with reinforcement bars.

Material Models

The following material models can be used in combination with embedded reinforcements:

- Linear Elastic
- Von Mises plasticity with optionally hardening diagrams
- Monti-Nuti model for cyclic loading conditions
- User-supplied subroutines

Post-tensioning

In post-tensioning the reinforcement is located in the mother element, but is not yet bonded to the concrete. This is the case when e.g. a tendon is located in a reinforcement channel, but not yet grouted. When post-tensioning, the reinforcement may be stressed without affecting the deformation of the concrete in which it is located. This situation can be achieved by labeling the reinforcement as **NOBOND**. In contrary to the default embedded reinforcement the **NOBOND** option has the following properties:

- The stiffness of the reinforcement does not contribute to the stiffness of the mother element,
- The deformation of the mother-element does not cause stresses of strains in the reinforcement,
- If a pre-stress is applied to a **NOBOND** reinforcement than the equivalent element-forces are applied as external forces to the element.

After the initial stresses have been applied to the un-bonded reinforcement, the reinforcement will be fully grouted (bonded) to the mother element. The bonding or no-bonding is defined as a material property in DIANA. To change from no-bonded reinforcements in earlier construction stages to bonded reinforcements in later construction stages, the **NOBOND** command can be included in the material definition initially and removed for later stages, or the user can define the **NOBOND** option in the material definition and define for which construction stages the **NOBOND** behavior must be replaced by bonding by the command **BOND** in the **EXECUT PHYSIC** command. This bonding is mostly associated with grouting the reinforcement channels. After bonding, the reinforcement contributes to the stiffness of the mother element and the reinforcement strains and stresses change upon deformation of the mother element.