Stretto di Messina

Concessionaria per la progettazione, realizzazione e gestione del collegamento stabile tra la Sicilia
e il Continente Organismo di Diritto Pubblico
(Legge n° 1158 del 17 dicembre 1971, modificata dal D.Lgs. n°114 del 24 aprile 2003)

Eurolink

PONTE SULLO STRETTO DI MESSINA

PROGETTO DEFINITIVO

EUROLINK S.C.p.A.

IMPREGILO S.p.A. (MANDATARIA)
SOCIETÀ ITALIANA PER CONDOTTE D'ACQUA S.p.A. (MANDANTE)
COOPERATIVA MURATORI E CEMENTISTI - C.M.C. DI RAVENNA SOC. COOP. A.R.L. (MANDANTE)
SACYR S.A.U. (MANDANTE)
ISHIKAWAJIMA - HARIMA HEAVY INDUSTRIES CO. LTD (MANDANTE)
A.C.I. S.C.P.A. - CONSORZIO STABILE (MANDANTE)

IL PROGETTISTA	IL CONTRAENTE GENERALE	STRETTO DI MESSINA	STRETTO DI MESSINA
COWI Ing. E.M. Veje Dott. Ing. E. Pagani Ordine Ingegneri Milano n° 15408 Eurolink	Project Manager (Ing. P.P. Marcheselli)	Direttore Generale e RUP Validazione (Ing. G. Fiammenghi)	Amministratore Delegato (Dott. P. Ciucci)

PF0068_F0

Unità Funzionale	OPERA DI ATTRAVERSAMENTO
Tipo di sistema	SOTTOSTRUTTURE
Raggruppamento di opere/attività	BLOCCHI D'ANCORAGGIO
Opera - tratto d'opera - parte d'opera	General
Titolo del documento	Semi-local FE Model Description, Annex

CODICE	CG1000	P	CL	D	P	ST	B4	00	00	00	00	01	F0

REV	DATA	DESCRIZIONE	REDATTO	VERIFICATO	APPROVATO
F0	20/06/2011	EMISSIONE FINALE	PFJ	GBA	IJ/ABI

NOME DEL FILE: PF0068_F0_ANX

INDICE

1 Introduction

This document gives a short presentation of the general parametric IBDAS Model used for the design and analysis of the Messina Strait Bridge Anchor Blocks. The document should also be seen as a guideline on how to read results.

IBDAS (Integrated Bridge Design and Analysis System) is a general computer aided structural design and analysis system developed by COWI. IBDAS is based on 3D parametric solid modelling and provides procedures for fully integrated design and analysis of bridges.

This report is structured as follows: Chapter 1 contains an introduction. In Chapter 2 a short description of the basis of the design is given. The used 3D geometry model is described in Chapter 3. Chapter 4 explains the material and verification model. In Chapter 5 the finite element model. Chapter 6 contains the load model and in chapter 7 the construction process model is presented.

In the Appendix A an overview of the geometry model, material and verification model, the finite element model and the load model of the Calabrian Anchor Block is given. Appendix B shows the same for the Sicilian Anchor Block.

2 Design Basis

Information from the design group, definition drawings and contract documents form the basis for establishing the geometry model, the material model, the finite element model, the load model and the construction process model for the anchor blocks. The resulting structural model (geometry- and material model) is used directly to generate general layout drawings, setting out data, quantities and selected 3D-pictures. This procedure secures that the structural model is in accordance with the outline and, thus, the basis for the finite element model is correct.

The general parametric IBDAS Model for the Calabrian Anchor Block is named ANCH_C and analogue ANCH_S for the Sicilian Anchor Block. On all output the anchor block name will be presented. The following general IBDAS models are used:

- **EC2_BR**: Eurocode 2, Part 2 (concrete bridges) and Eurocode 3, Part 2 (steel bridges):

  EC2, Part 2 version: EN 1992-2:2005

  EC3, Part 2 version: EN 1993-2:2006

  The actual material model (MATE) is obtained by customisation of this-standard code of practice model.

- **CATALOG**: IBDAS Model system defining steel characteristics (fabrication data) for different prestressing systems. The IBDAS CATALOG Model is described in Chapter 2 in the IBDAS manual "Code of Practice System Database".

The global coordinate system is an orthogonal left hand SYZ-coordinate system where the S-axis is horizontal following the longitudinal direction of the bridge. The Z-axis is vertical positive upwards. The Y-axis is horizontal. For both anchor blocks the S-axis is oriented toward the centre of the bridge. The points of origin are placed at Z-level 0.00m directly below the theoretical cable point at the splay saddle.

3 Geometry Model

Each anchor block is modelled geometrically correct in order to form the base for the construction drawings and to achieve the correct structural volume (dead load) and stiffness, which in IBDAS automatically are computed based on the geometric input. A documentation of the geometry model for each anchor block is given in the Appendix A1 and B1.

Since the purpose of the model is the global analysis and verification of the anchor block minor simplifications of the geometry are made:

The anchor chamber roof is model as plate with constant thickness. Due to this the right stiffness and stress distribution is ensured in the model. The design of the roof with prefabricated beams is based on a separate sub-model.

Furthermore secondary structural elements are neglected in the model, for example the access to the anchor chamber and the soil retaining wall at the back of the block. The members are designed separately.

The pre-stressing loops are arranged in groups of one to four tendons (represented by RGRP1-4). Each group is modelled as one resulting tendon with the according cross-section of the group. Each group is defined by the parametric IBDAS model called PT_loop with the following model number:

PT_loop_model_no = 4000 + 1000*cable + 200*side + 10*layer + loop

where:

cable: position in relation to CL-anchor block (=1: left; =2: right)

side: position in relation to CL-cable plane (=1: left; =2: right)

layer: layer no 1-10 corresponding to layer A, B, …, J

loop: loop no 1-6

Due to the intended casting sequence with rectangular blocks the inclined surfaces of the anchor blocks consists of several steps. However, the stepped surfaces are modelled as smooth surfaces.

4 Material Model

Regarding the structural analysis linear elastic material behaviour is assumed. For the verification the material nonlinearity is taken into account.

The material model defines the necessary material properties for all materials used in IBDAS calculations together with definition of adequacy verification procedures for concrete structures. The material model used for both anchor blocks consists of one input file named mate.min. This file is listed in the appendix A2.

The following materials are used in the actual IBDAS models.

4.1 Prestressing Tendons Properties

The given friction coefficients μ and k are in relation to the following formula:

$$P_x = P_0 \cdot e^{-(\mu \cdot \alpha + k \cdot x)}$$

The following prestressing cable type is defined:

PSTEEL201: Prestressing tendons EN10138-3 Y1860S7-15.7 with 19 strands of 150mm^2.

Nominal strand diameter	15.7 mm
Nominal strand area	150 mm^2
Nominal weight of strand	1.18 kg/m^3
Nominal ultimate tensile strength f_{uk}	1860 MPa
Characteristic modulus of elasticity	195000 MPa
Initial tension	0.75 f_{uk}
Drawin	8 mm
Friction coefficient (μ)	0.2 /rad
Wobble factor k (internal tendons):	0.002 /m

Relaxation after 1000H is defined according to EC2 figure 4.8 class 2 (low relaxation strands).

4.2 Reinforcement Steel Properties

The following type of reinforcement steel (hot worked) is defined:

RSTEEL1: Reinforcement steel with f_{yk} = 450 MPa

Characteristic modulus of elasticity: 200000 MPa

4.3 Structural Concrete Properties

The following types of structural concrete are defined:

MATE1/MATE2/MATE3: Concrete C30/37(f_{ck} = 30 MPa)

Density of reinforced concrete: 0.025 MN/m^3

Coefficient of thermal expansion α: $1.0 \cdot 10^{-5}/^{o}C$

Poisson's ratio ν: 0.2

Creep and shrinkage according to EC2, part1 with short time modulus of elasticity equal 32837 MPa.

4.4 Structural Adequacy Verification

Verification of structural adequacy for each design action case (each criterion) in each stress point according to the actual design requirements are obtained via use of the verification models included in the IBDAS MATE model.

A short description of verification principles for three dimensional stress state in solid elements is given below:

The point stresses used are the total stresses found in service analysis. This stress state is kept constant for the verification. Hence, the approach does not capture any stress redistribution capacities. In doing so, the design procedure is on the safe side. The specific point strain state $(\varepsilon_{ss},\varepsilon_{yy},\varepsilon_{zz},\gamma_{sy},\gamma_{sz},\gamma_{yz})$ which corresponds to the actual point stresses $(\sigma_{ss},\sigma_{yy},\sigma_{zz},\sigma_{sy},\sigma_{sz},\sigma_{yz})$ considering the design stress-strain for solid material and applied reinforcement distributed as 3D-mesh (defined via postprocessing reinforcement) is found. The uniaxial design stress-strain relationship for solid material is assumed to apply in each principal strain direction just extended with a reduction of compression strength at simultaneous tension strain in the any other direction matching EC2, appendix F (υ factor as Collin reduction). The strain state is superimposed (if desired) by an initial strain state which considers the construction process and/or short term deformation actions (temperature etc.) introduced initial stress state. The determination of the strain field, which just provides the actual sectional forces, is at non-linear stress-strain relationships a nonlinear problem which basically is attempted to be solved by a Newton-Raphson iteration (same procedure as at beam investigation). The iteration procedure comprises the following steps:

1. The solid material and each unidirectional reinforcement mesh is included separately

2. The initial values of material elasticity are used to assemble the stiffness matrices for the solid material and steel.

3. The stiffness matrices are added to produce the initial local point stiffness matrix **G**.

4. The generalized strain vector $\bar{\varepsilon} = (\varepsilon_{ss}, \varepsilon_{yy}, \varepsilon_{zz}, \gamma_{sy}, \gamma_{sz}, \gamma_{yz})$ is found from the equation:

   $$\mathbf{G} \cdot \bar{\varepsilon} = \bar{\sigma}$$

   where $\bar{\sigma} = (\sigma_{ss}, \sigma_{yy}, \sigma_{zz}, \sigma_{sy}, \sigma_{sz}, \sigma_{yz})$ is the generalized stress vector.

5. The strains in solid material are resolved into principal directions.

6. Stresses in solid material are calculated using the actual stress/strain diagram.

7. Strains and stresses in each reinforcement layer are calculated. Only reinforcement layers distributed as 3D-mesh (defined via postprocessing reinforcement) are included.

8. Stresses are integrated into a resisting point stress vector $\bar{r}$.

9. It is checked whether $\bar{r}$ corresponds to $\bar{\sigma}$. If convergence (or iteration limit exceeded) go to step 14.

10. The tangent modules for solid material and each reinforcement mesh are calculated and combined together to produce the total tangential stiffness matrix **T**.

11. The equation:

    $$\mathbf{T} \cdot \Delta\bar{\varepsilon} = \bar{\sigma} - \bar{r}$$

    is solved.

12. $\Delta\bar{\varepsilon}$ is added to $\bar{\varepsilon}$.

13. Repeat from step 5 above.

14. Reporting on stresses, strains, etc. or on lack of convergence.

Neither the reinforcement percentages nor their directions are changed during the iteration process.

The procedure for determination of the stress state for a known strain state, described above, is also used in a limit state analysis as the cross section model.

At plotting of effects, the design resistance is presented via the following utility ratios:

Reinforcement tension utility ratio:

$$\mathbf{ur\_reinf\_tens} = \frac{\sigma_{s,\max}}{f_{ttd}}$$

Concrete compression utility ratio

$$\mathbf{ur\_conc\_comp} = \frac{\sigma_{c,\max}}{f_{cd}}$$

Other effects reported:

c_ps1 / c_ps2 / c_ps3: principal stresses.

e_ps1 / e_ps2 / e_ps3: principal strains.

c_ps_max / c_ps_min : max/min principal stress.

e_ps_max / e_ps_min : max/min principal strain.

In case principal stresses/strains are reported as vector plot, both max and min are shown.

Appendix A6 and B6 contains selected output from analysis and verification.

5 Finite Element Model

5.1 Discretisation

The finite element models to be used are 3-D finite element models built up by the implemented system of parabolic isoparametric solid elements (IBDAS SOLID60 elements) with reduced numerical integration.

This ensures:

- A generally recognized, reliable system of elements providing very exact results in the integration points, even when the structure is sub-divided into a coarse mesh of elements.
- A system of elements shaped to represent irregular, curved geometrical patterns. Structures with curved surfaces therefore are modelled geometrically correct.
- Realistic visualizations of analysis models in undeformed as well as deformed shapes.

The finite element models are defined logically in relation to the 3-D structural model. The definition requires very little input. The finite element model generation includes:

- Automatic generation of nodes, element topology and cross section constants.
- Analysis elements including user-selected reinforcement. All analysis element types accept reinforcement and those parts of the reinforcement, which lie within individual analysis elements, are automatically identified.
- Possibility for user-specified logical deviations between the 3-D structural model and the finite element model. In this way analytically irrelevant structural details are ignored.
- Possibility for definition of multilevel super-element assemblies.

The finite element model for each anchor block is a 3-D finite element model built up by parabolic isoparametric solid elements (IBDAS SOLID60 elements) with reduced numerical integration.

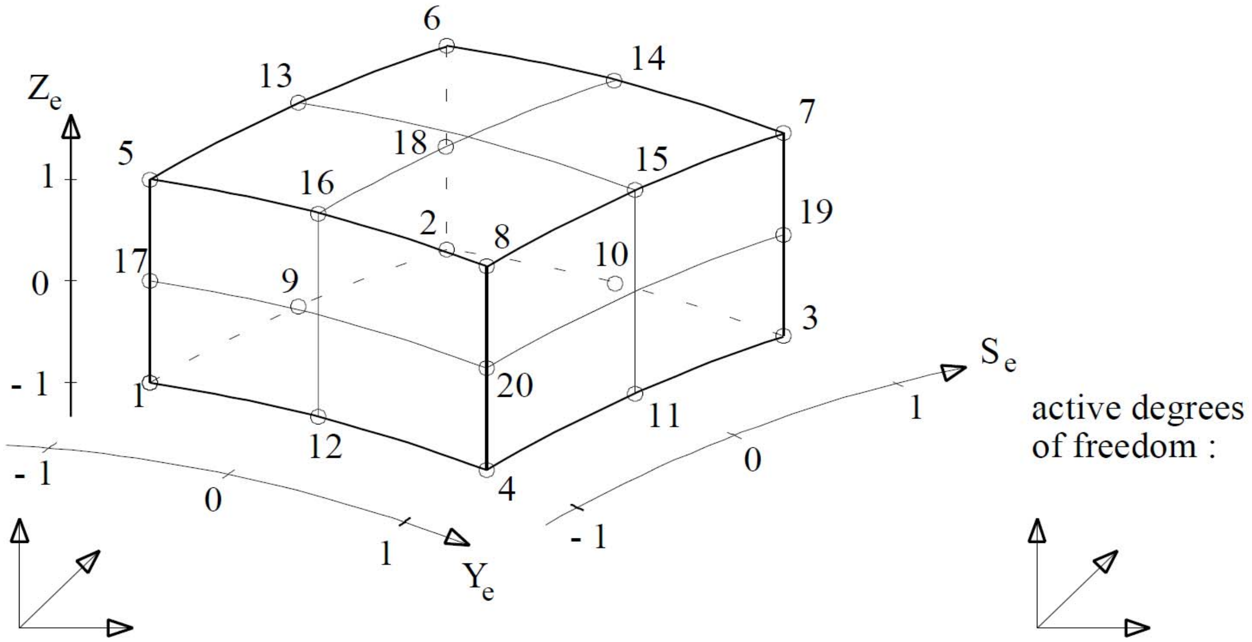


Figure 1: General isoparametric solid element SOLID60.

The local orthogonal finite element coordinate system (Se,Ye,Ze) is defined by:

Se: Intersection between surface parallel with top face/bottom face of layer (isoparametric interpolation) and surface parallel with the LCUT's isoparametric interpolation). Orientation in direction of increasing TCUT numbering. Se is a dimensionless coordinate [-1,1].

Ye: Situated in surface parallel with top side/under side of layer and perpendicular to the Se-direction. Orientation in the direction of increasing LCUT numbering. Ye is a dimensionless coordinate [-1,1].

Ze: Determined by that the Se,Ye,Ze-system shall be an orthogonal coordinate system of the same type (right/left) as the finite element model coordinate system. Ze is a dimensionless coordinate [-1,1].

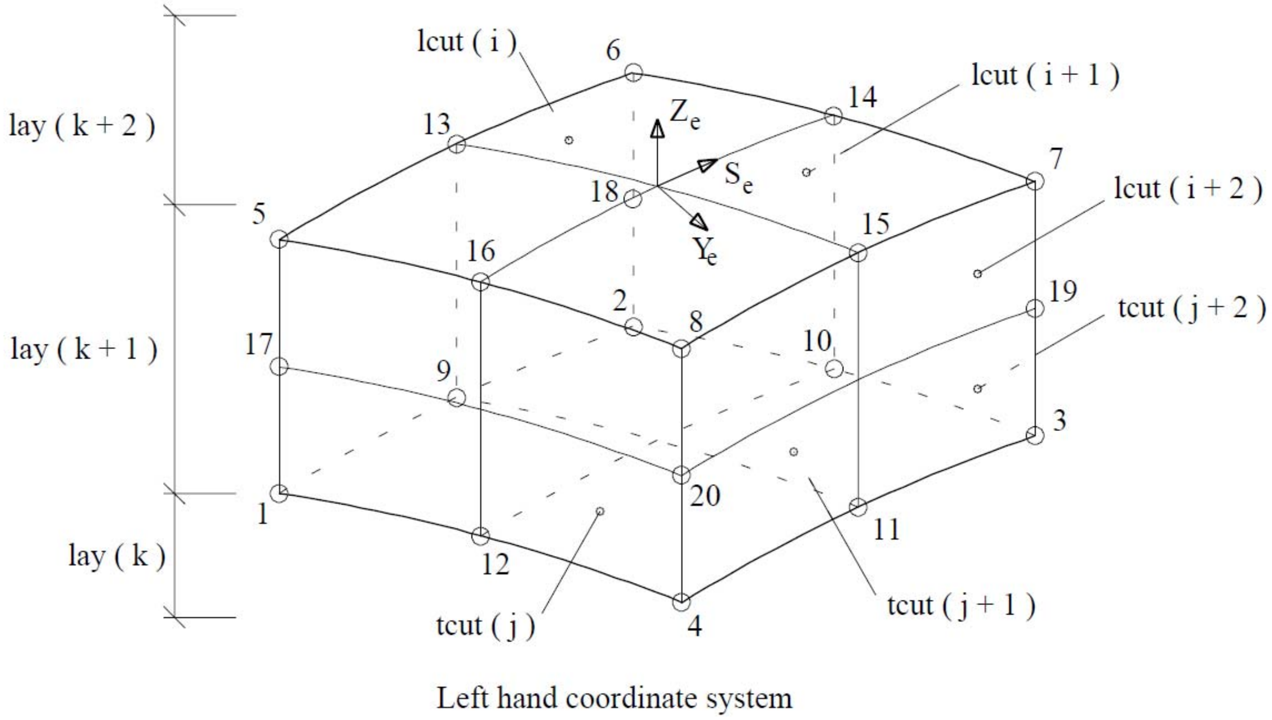


Figure 2: Local coordinate system for spatial SOLID60 element

The solid element has 8 Gauss points (integration points) located in the corners relatively 0.21 from each side surface. Element stiffness matrices are calculated via reduced Gaussian numerical integration.

The signs for the stress resultants in an infinitesimal element around a gauss point in a solid element are shown in the figure below for a left hand coordinate system. The analysis model and, thus, the results are based on the left hand coordinate system. It should, however, be noted that it generally applies that normal stresses are taken to be positive when they act outward, i.e., when they are tensile stresses. A shear stress component is taken to be positive when the component on an element with normal in the direction of a positive, or negative, axis also has the direction of a positive, or negative, axis, respectively.

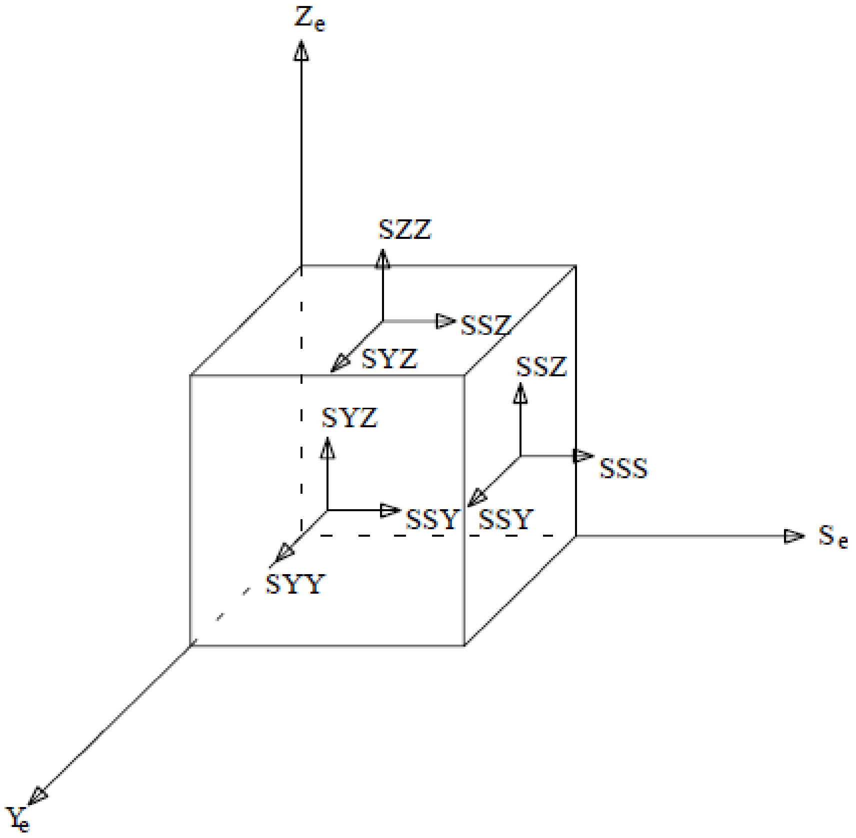
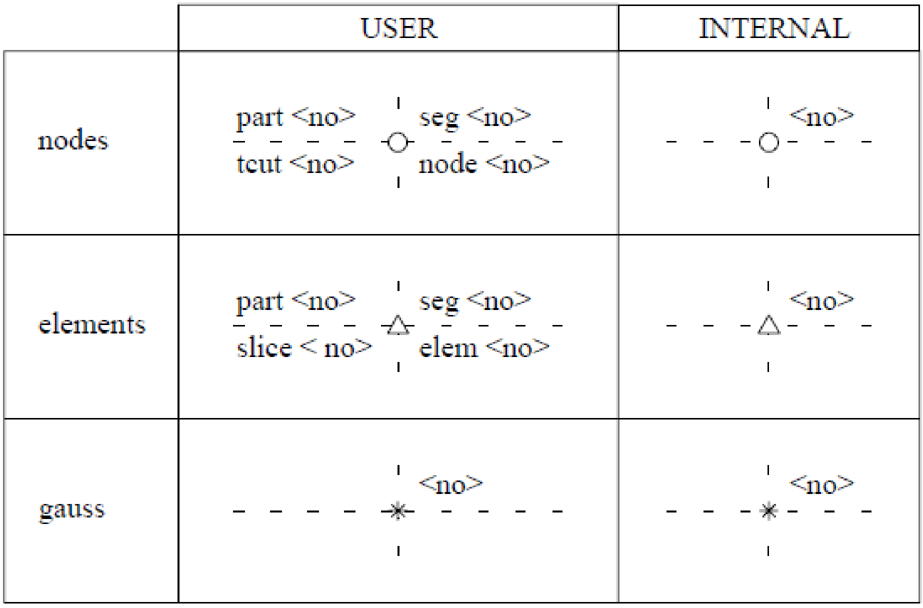


Figure 3: Stress components at stress point for a SOLID60 element. Infinitesimal element around stress point shown.

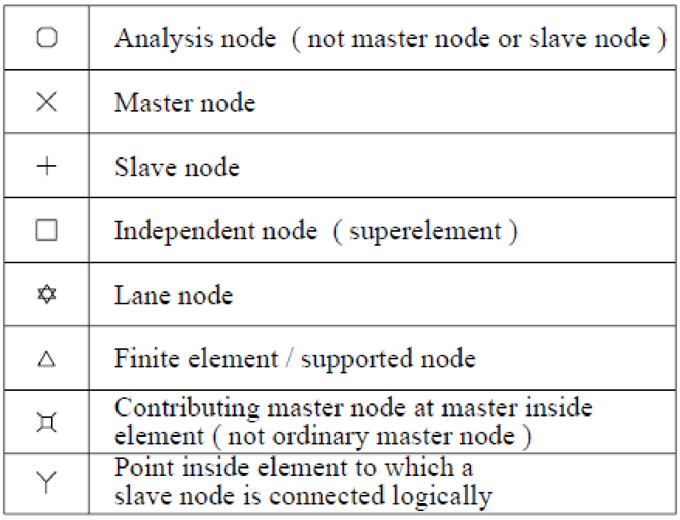
IBDAS uses a numbering system where the actual structure is divided into parts, each part into segments, each segment into slices and each slice into elements.

Each finite element is consequently identified by the actual part no., segment no., slice no. and element no. The figure below shows how the element numbering appear on drawings and explains the symbols used.



	USER	INTERNAL
nodes	part <no> tcut <no> ○ seg <no> node <no>	○ <no>
elements	part <no> slice < no> △ seg <no> elem <no>	△ <no>
gauss	✳ <no>	✳ <no>

Figure 4: Numbering of Gauss points, nodes and elements in finite element model.



○	Analysis node (not master node or slave node)
×	Master node
+	Slave node
□	Independent node (superelement)
✡	Lane node
△	Finite element / supported node
⌑	Contributing master node at master inside element (not ordinary master node)
Y	Point inside element to which a slave node is connected logically

Figure 5: Symbol table.

5.2 Boundary Condition

During all the construction phases (until phase 100: opening of bridge) the anchor block is supported by horizontal and vertical spring supports distributed uniformly across the bottom surface

Vertical spring stiffness: kz = 30 MPa/m

Horizontal spring stiffness: ks = ky = 15 MPa/m

At opening of bridge (phase 100) and later phases the spring supports are re-moved and the structure is now in each load case supported by an earth pres-sure distribution (normal pressure and shear) found via ABAQUS calculations (see CL-D-P-ST-B4-BS-00-00-00-01 and CL-D-P-ST-B4-BC-00-00-00-01) and adjusted to secure full equilibrium.

6 Load Model

The loads of the anchor saddle, the cable forces, the fill loads and the soil pressure are all applied as surface loads on the according solid elements. A visualisation of the element mesh and applied loads for all basic load cases are given in the appendix A4 and B4. The cable forces are delivered from the main IBDAS bridge model. Besides the cable force from dead load six SLS load cases and six ULS load cases are investigated. For both, SLS and ULS, five earthquake scenarios are investigated. The earthquake load on the anchor block itself is modelled as volume load following from the unit weight and the accelerations. The Appendix contains an identification table for the complete load system and the load combinations A4 and B4.

7 Construction Process Model

A construction process is in IBDAS modelled as a sequence of construction phases each phase consisting of activities such as: Casting, building in and removing structural parts, prestressing, injection and slackening of prestressing tendons, changing support/coupling conditions and placing and removing temporary construction load.

A time indication is linked to each phase with special reference to the calculation of time dependent effects such as: Creep, relaxation and shrinkage.

All activities associated to a given construction phase are analytically treated as being simultaneous.

The Sicilian Anchor Block is built in 6 construction phases, and the Calabrian Anchor Block is built in 4 construction phases. In each phase one layer is built in. This is an appropriate modelling of the real construction process. One layer represents a number of casting blocks. After building in all layers the prestressing is activated (Phase 24 for Calabrian, Phase 26 for Sicilian) and finally the anchor chamber roof is installed. Phase 100 reflects the opening of the bridge where the cable load is applied. The spring supports are removed and the structure is supported by the according earth pressure distribution with normal pressure and shear determined by ABAQUS. Construction phase 200 represent 200 years after opening of the bridge.

Report of all activities are contained in appendix A5 and B5.

Appendix A : Calabrian Anchor Block

A1 Geometry Model

A2 Materials and Verification Models

A3 Finite Element Model

A4 Load Model

A5 Construction Process Model

A6 Verification

A1 Geometry Model

A2 Materials and Verification Models

A3 Finite Element Model

A4 Load Model

A5 Construction Process Model

A6 Verification

Appendix B : Sicilian Anchor Block

B1 Geometry Model

B2 Materials and Verification Models

see Appendix A2

B3 Finite Element Model

B4 Load Model

B5 Construction Process Model

B6 Verification

B1 Geometry Model

B2 Materials and Verification Models

see Appendix A2

B3 Finite Element Model

B4 Load Model

B5 Construction Process Model

B6 Verification