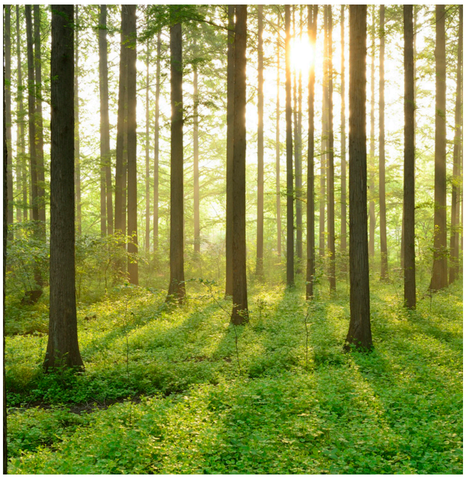
GUIDELINE FOR FE ANALYSES OF CONCRETE DAMS

REPORT 2016:270



HYDROPOWER

Energiforsk

Guideline for FE analyses of concrete dams

RICHARD MALM

ISBN 978-91-7673-270-0 | © 2016 ENERGIFORSK

Energiforsk AB | Phone: 08-677 25 30 | E-mail: kontakt@energiforsk.se | www.energiforsk.se

Förord

Målet med projektet var att sammanställa en beskrivning över hur numeriska analyser bör genomföras för dammkonstruktioner, med fokus på betongdammar. Ett annat mål var att sprida kunskap om tillämpningar, möjligheter och begränsningar med finita elementmetoden som ett analysverktyg för SVCs intressenter.

Detta projekt har bedrivits inom Svenskt VattenkraftCentrum (SVC). SVC är ett centrum för utbildning och forskning inom vattenkraft och gruvdammar. SVC har etablerats av Energimyndigheten, Energiforsk och Svenska Kraftnät tillsammans med Luleå tekniska universitet, Kungliga Tekniska Högskolan, Chalmers tekniska högskola och Uppsala universitet.

Medverkande organisationer är: Andritz Hydro, Falu Energi & Vatten, Fortum Generation, GE Renewable Energy, Holmen Energi, Jämtkraft, Jönköping Energi, Karlstads Energi, Mälarenergi, Norconsult, Rainpower, Skellefteå Kraft, Sollefteåforsens, Statkraft Sverige, Sweco Energuide, Sweco Infrastructure, SveMin, Umeå Energi, Uniper, Vattenfall Research and Development, Vattenfall Vattenkraft, VG Power, WSP Samhällsbyggnad och ÅF Industry.

Sammanfattning

Numeriska 3D analyser baserade på finita elementmetoden (FE-metoden) ger möjlighet att genomföra mer noggranna och detaljerade analyser jämfört mot motsvarande traditionella dimensioneringsmetoder. Finita elementanalyser tillämpas idag vanligen för utvärdering, dimensionering etc. av betongdammar. Denna utveckling är dock inte enbart utav godo. Dessa numeriska analyser och modeller kräver en gedigen teoretisk kunskap om tillämpbarheten av dessa metoder dels från utföraren, men även från de som granskar analyserna. Dessutom, krävs att resultaten från beräkningarna utvärderas noggrant baserat på underliggande antaganden och dess praktiska relevans.

Syftet med föreliggande rapport är att kunna användas som en riktlinje för ingenjörer som genomför strukturmekaniska finita elementberäkningar på betongdammar.

I denna rapport behandlas följande aspekter som beräkningsingenjörer ställs inför då de ska genomföra strukturmekaniska analyser av betongdammar

- När kan/bör FE-analyser tillämpas och även ge exempel på tillämpningar på vad dessa typer av beräkningar kan användas för
- Definition av geometrisk modell
- Definition av elementnät (diskretisering av modellen)
- Definition av materialegenskaper och beteende
- Definition av randvillkor och kontaktvillkor
- Definition av statiska, miljöbetingade (temperatur och fukt) samt dynamiska (seismiska) laster
- Lösningsmetodiker för strukturmekaniska analyser
- Utvärdering och verifiering av resultat
- Säkerhetsformat

I efterföljande kapitel ges rekommendationer avseende modelleringsaspekter, materialegenskaper etc. Dessutom, ett kapitel behandlar även praktiska modelleringstips för några vanligt förekommande fall där t.ex. förslag ges på metoder som kan användas för att extrahera snittkrafter, genomföra stabilitetsanalyser (brottanalyser) etc.

Tanken med denna rapport är att den ska vara oberoende av val av programvara för att genomföra finita elementberäkningarna. Orsaken till detta är för att denna riktlinje på så sätt kan tillämpas av en större målgrupp oberoende av vilken programvara som tillämpas. Metodikerna som beskrivs i denna rapport presenteras därför på ett generellt sätt så att de ska gå att tillämpa i de flesta typer av FE-programvaror

Summary

Numerical 3D finite element (FE) analyses allows the engineers to perform more accurate and detailed analyses, compared to the traditional design methods. Today, FE-analyses are a common tool for assessment, design etc. of concrete dams. However, there are not only positive aspects of this development. These numerical analyses require a solid theoretical background of the applicability of methods, both from the FE-engineer performing the analyses but also for the reviewers of the results. In addition, the results obtained requires careful interpretation with respect to the underlying assumptions and their practical relevance.

The intention of this report is to serve as a guideline for engineers performing finite element (FE) analyses of the structural (mechanical) behaviour of concrete dams.

In this report, the different aspects that a FE-engineers are faced when defining models and analysing concrete dams have been described such as

- When may FE analyses be required and examples of what these models be used for
- Definition of the geometrical model
- Definition of the element discretization
- Definition of the material behaviour
- Definition of boundary conditions and constraints
- Definition of static, environmental (temperature and moisture) and dynamic (seismic) loads
- Solver solution techniques
- Evaluation and verification of the results
- Safety formats

In the following chapters, recommendations regarding modelling aspects, material properties etc. are given. In addition, one chapter also contains practical modelling tips for some common cases where for instance suggestions to extract sectional forces, define push-over analyses etc. are given.

The intention of this report is that it is unrelated to the choice of software used. The reason for this is that the guideline should be able to use regardless of choice of finite element software. The methodology given in this report is defined in general terms, and could thereby be implemented in most FE codes.

List of content

Energiforsk

1 Introduction

Numerical analysis, based on the finite element (FE) or the finite difference (FD) method, is a powerful tool for assessment, design, etc. of concrete dams. Within the field of structural engineering, finite element (FE) analyses are today a very common tool, and the use of 3D finite element analyses has increased significantly over the last years, (Pacoste et al. 2012).

The finite element method has been used for more than 60 years. However, it is the vast development of computer hardware and software over the last decades has made it possible for computer-based methods to be widespread as tools, as they are today. The technical development has led to a significant change of the civil and structural engineering sector where tools such as 3D computer aided-design (CAD), building information modelling (BIM) and finite element analyses (FEA) are part of most design projects today. Different fields of civil and structural engineering, houses, bridges, dams etc. has implemented these computer-based fields to varying extent, where today (in Sweden) BIM is highly integrated in many building projects and 3D FE-analyses is almost a requirement for design of bridges. It is likely to believe, that this development will progress even further during the coming years.

Numerical 3D finite element analyses allows the engineers to perform more accurate and detailed analyses, compared to the traditional design tools. However, there are not only positive aspects of this development. These numerical analyses require a solid theoretical background of the applicability of methods, both from the FE-engineer performing the analyses but also for the reviewers of the results. In addition, the results obtained requires careful interpretation with respect to the underlying assumptions and their practical relevance. Besides this, one other concern is regarding how the results can be stored in order to be used in the future, for instance when todays structures needs strengthening or service life extending measures. Some software's can only save graphical user files and it is questionable that these files will be possible to access several years from now.

1.1 AIM AND SCOPE

The aim of this report is to be a guideline for structural engineers when performing structural (mechanical) finite element analyses of concrete dams. The aim of this report is to assist the FE-engineers with answers to following questions that may arise during a project

- When is it suitable to use FE-analyses for mechanical analyses of concrete dams?
- What kind of numerical model/analysis is needed?
- What would be the benefit from using FE-analyses for mechanical analyses of concrete dams?
- How should the numerical analysis be performed?

The intention of this report is that it is unrelated to the choice of software used. The reason for this is that the guideline should be able to use regardless of choice of finite element software. The methodology given in this report is defined in general terms, and could thereby be implemented in most FE codes.

1.2 LIMITATIONS

In this guideline, recommendations are given regarding of modelling of concrete dams. This report is focused on structural engineering analyses and hence, other topics related to hydropower and dams are not considered.

Other concrete structures, for instance in a hydropower facilities, may also benefit from the recommendations given in this report. However, these structures may also be subjected to other types of loads that are not considered in this report.

One other limitation is that, despite that the techniques presented in this report may be used regardless of country, the presented material properties are based on Eurocode and the load values given in this report are based on the Swedish design guideline RIDAS (2011).

This report does intentionally not include the aspects regarding review of FE-analyses. This topic is covered by another Energiforsk report by Ekström et al. (2016). This report is focused on review of numerical analyses of concrete dams and is a good complement to this report.

2 Numerical analyses of dams

Numerical analyses of hydraulic concrete structures are used in research, design of new dams, assessment of existing dams etc. In dam engineering, numerical analyses are mainly used for

- Mechanical analyses
- Transport analyses
- Fluid mechanics analyses

This report focuses on mechanical analyses i.e. mechanics on structural and material level (deformations, stresses, load capacity, cracking etc.) and to some extent on transport analyses such as temperature and moisture. Transport analyses, such as mass transport of chemicals in the concrete etc. and fluid mechanics analyses are out of the scope of this report.

In this section, some examples of cases when the use of numerical analyses can be of great benefit for a project are given.

2.1 EXAMPLE OF CASES WHEN NUMERICAL ANALYSES ARE NOT NEEDED

It is important to point out that the use of finite element analyses is not necessary in all projects. If the traditional design methods are sufficient to evaluate a dam, then there is no need for more detailed analyses. One example where traditional design methods often may be sufficient is conventional stability analyses of gravity dams. This is the case, if the dam is expected to act as a monolithic structure (i.e. no cracks occurs that could result in internal failure modes) and that the dam (with conservative assumptions) satisfies the stability requirements with sufficient margins. If it is deemed that numerical analyses would not result in any additional information of importance, then it is sufficient to use traditional stability analyses.

As shown by Fu and Hafliðason (2015), when using the load concept later described in Section 12.2, numerical analyses gives almost identical results as the traditional analyses. The case study analysed by Fu and Hafliðason (2015) is illustrated in Figure 2-1 and the results from are summarized in Table 2-1.

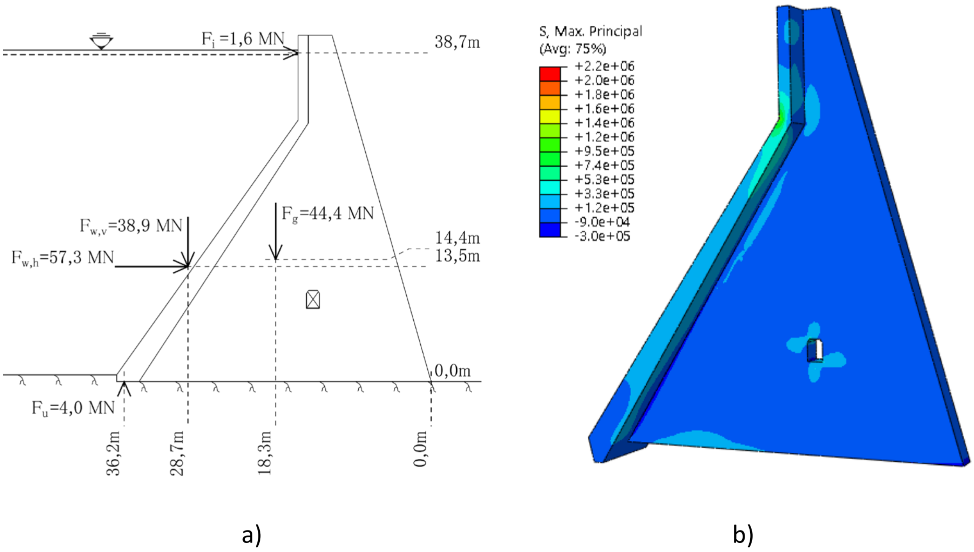


Figure 2-1 Sketch of geometry and loads considered by Fu and Hafliðason (2015).

Table 2-1 Safety factors for an uncracked buttress dam, obtained from traditional stability calculations and numerical analyses

	Sliding failure	*Overturning failure*
Traditional stability calculation	1.35	1.97
3D numerical analysis	1.33	1.94

It should be noted, that (in this case) the numerical analyses showed slightly lower factors of safety. This is often the case and for instance, another example of this is presented in Section 2.2.2. However, this may not always be the case and some differences in the obtained safety factor may occur depending on modelling choices and underlying assumptions in both traditional design analyses and FE-analyses.

2.2 EXAMPLE OF CASES WHEN NUMERICAL ANALYSES MAY BE NEEDED

In this section, a few examples are presented where FE-analyses have played an important role in the project and where the models have been used as tools for decision-making.

2.2.1 Complicated geometry and/or loadings

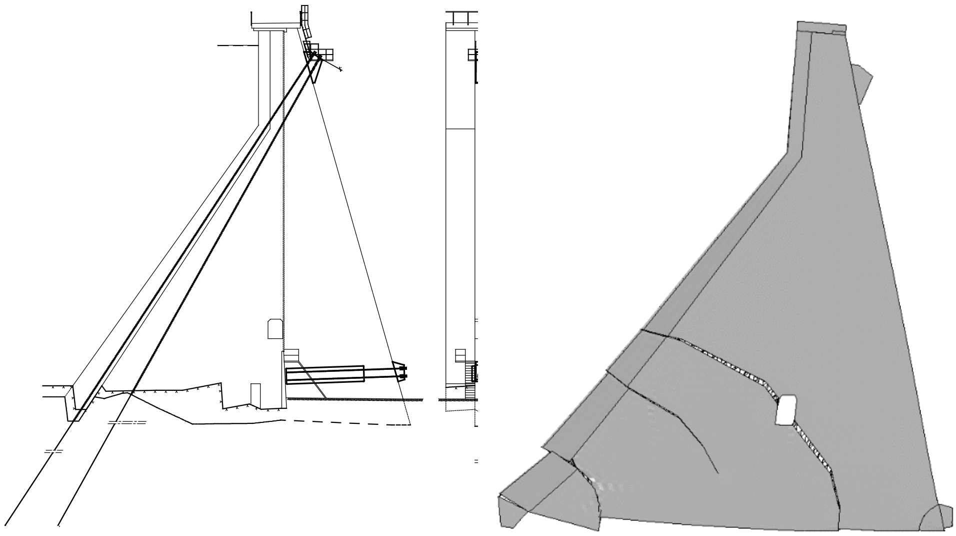
One obvious reason when finite element analyses should be used is if the geometry or loading is complicated to analyse with simple methods. For instance, the response of arch dams is often determined with means of finite element analyses. Their complex geometry and the fact that they can carry the loads horizontally by arch action in combination with vertically as cantilevers depending on the shape and design of the dam.

Another case that is more or less always analysed with the means of finite element analyses is seismic response of dams or other hydraulic structures. The reason is that simple methods often leads to rather conservative results.

2.2.2 Stability analyses

If the traditional stability analyses shows that, the requirements cannot be satisfied or if cracks are believed to compromise the stability of the dam or if the dam is strengthened, then finite element analyses may be beneficial.

This is for instance the case in Malm et al. (2016a), where a cracked dam with installed post-tensioned tendons. In this study, it was shown that the behaviour of the strengthened dam was not accurately captured and that the factor of safety was significantly underestimated in the traditional analyses.

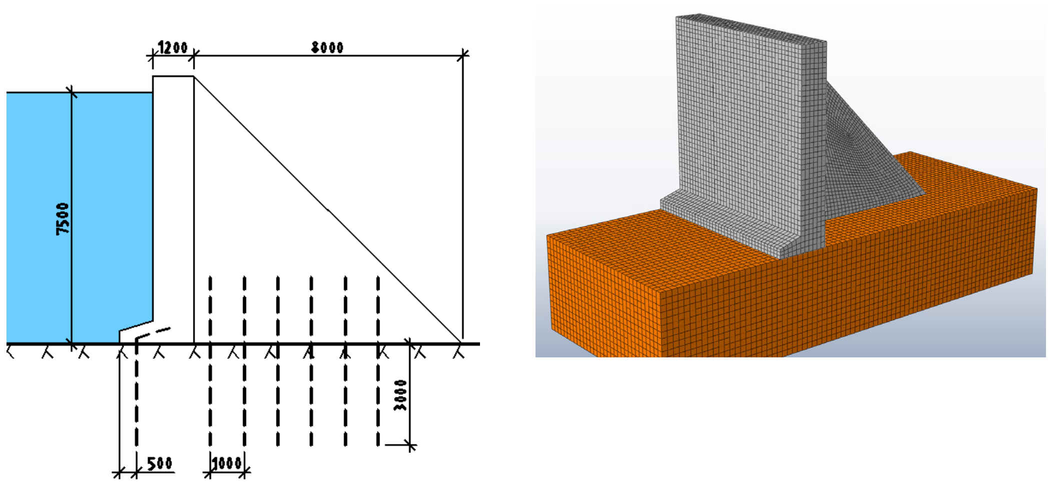


a) b)

Figure 2-2 Example of a cracked dam which has been strengthened, where numerical analyses where used. From Malm et al. (2016a).

Cracks influence the behaviour of dams in different ways, such as reducing the leak-tightness of the dam and increasing the hydraulic pressure within the material/structure. Furthermore, cracks may have an impact on the stiffness and stability of the dam. The ordinary sliding and overturning stability analyses are not sufficient when the supporting structure is cracked. The cracks may comprise the integrity and the homogeneity of the structure. A cracked, or for that matter even a repaired structure, cannot be regarded as a homogenous structure and should be treated accordingly. Therefore, other types of analyses are required instead of the traditional design methods for the stability analyses of the cracked and repaired dams. A cracked concrete structure is also more likely to be damaged by environmental agents, which could lead to further cracking or widening of the existing cracks. Therefore, the present conditions of many concrete dams needs to be evaluated. The type of cracks, causes for cracking and their influence on the dam safety may differ between the dams. (Malm et al 2013)

Another example where the traditional stability calculations are not sufficient is from Malm et al. (2016b), where a frontal slab buttress dam just barely fails to satisfy the sliding criteria. In this case, the numerical analyses shows significant lower safety factor for failure. The reason for this is that the failure mode obtained in the FE analyses is a combination of overturning and sliding. As the upstream toe of the dam is lifted of the ground, the contact area between the dam and the foundation is reduced. Thereby, the total shear force has to be transferred over all smaller area, resulting in higher shear stresses and as a consequence, in a lower factor of safety. It should be noted that it is common that the sliding and overturning failure modes are only theoretical, and that a real failure is expected to be a combination of them. In these cases, the safety factor obtained for the combined mode is lower than both the sliding and overturning failure modes respectively.



a) b)

Figure 2-3 Example of a buttress dam where the FE-analysis shows significantly reduced factor of safety for a combined failure mode. From Malm et al (2016b).

These examples show that the traditional stability analyses may not be sufficient and it is up to the engineers to assess the risk of potential failure modes that the underlying assumptions in the traditional stability analyses may not capture. As these examples showed, the factor of safety obtained from these traditional stability calculations can be conservative or non-conservative compared to, more detailed, 3D FE-analyses.

2.2.3 Dam surveillance

One strong advantage with FE-analyses is that they can be used as input for dam surveillance. The numerical models can be used in different aspects of dam measurements, for instance to determine suitable type of sensors, their placement and alarm values for the sensors. An example of this was, for instance, presented by Nordström et al. (2015) and Malm et al. (2016a).

Otherwise, the most common application of FE-analyses regarding dam surveillance is to verify the response of the dam. Dam measurements, only give results in isolated points, i.e. at the locations where the sensors are placed. However, it should be mentioned that in most cases, these sensors are placed at sections that have been identified as most critical. Regardless of this, it may be difficult to actually understand what is happening in a dam based solely on measurements.

One disadvantage with most measuring equipment is that they can only give results from an unknown deformed state, i.e. from the date the sensors were installed. In most cases, the instrumentation have been installed several years after the dam was taken into operation. By the use of numerical models that have calibrated and verified against available measurement data, it is possible to get better understanding of the dam response. It is also possible to use the model to extrapolate into certain events or scenarios.

This have been performed in many different projects, a few examples are Malla and Wieland (1999), Malm (2009), Andersson and Seppälä (2015), etc.

2.2.4 Construction process

There are many cases in the literature where FE-analyses have been used to analyse the construction process and the behaviour of young concrete in concrete dams. The reason for this is due to the massive cross-sections in dams and other hydraulic concrete structures. Conventional building design codes were not developed for cross-sections as massive as those in dam engineering. The cross-sectional thickness of bridges, houses etc. are typically less than one meter, while hydraulic structures may have several meters thick cross-sections. Some examples of case studies where young concrete for hydraulic structures have been analysed are; James and Dollar (2003), Rueda et al. (2005), Ishikawa (1991), Kölfors (1994), Chen et al. (2001), Messer et al. (2011), Niu et al. (1995), Castilho (2003), among others.

As it can be seen in the references mentioned above, FE-analyses have successively been used to analyse the behaviour of young concrete for the hydropower industry in many research and design projects. Among these references, some are using simplified methodology based on design codes (which is discussed in Section 5.1.3) while others used more advanced methods that includes the coupled behaviour of heat, moisture and strength increase (which is discussed in Section 7.2.2).

2.3 PERFORMING LINEAR VS NONLINEAR ANALYSES

There are different types of nonlinearities that could influence the response of a concrete hydraulic structure. The nonlinearity may originate from the following

- Boundary or interaction nonlinearity
- Material nonlinearity
- Geometric nonlinearity

Boundary or interaction nonlinearity is due to nonlinear response of the dam due to joints, cracks in concrete or fractures in the rock and may play a significant role in the behaviour of the dam under serviceability conditions (i.e. for normal loads). However, as the intensity of loads increase and becomes close to the capacity of the dam, the larger impact these will have. Nonlinear boundary conditions may also originate due to settlements, seismic loads etc.

In general, for concrete dams, nonlinearities due to existing cracks and contraction joints are usually activated first as soon as the loads are sufficient to mobilize the friction in the cracks or to open these cracks. Nonlinear behaviour of the contraction joints is activated as soon as the load intensity is sufficient to break the bond in these joints. As the load intensity continues to increase, material nonlinearity such as initiation of new cracks or yielding of reinforcement in existing cracks may also occur. Finally, as the strains and displacements are further increased, the general assumption of small displacements may no longer be valid and hence second order effects (geometric nonlinearity) may have to be considered. These second order effects are however, rarely of great importance and may be neglected in most cases.

In design, linear analyses should be used as far as possible. It is not reasonable to use nonlinear analyses to a large extent in design because the required effort is significantly higher and the cpu-time is several times longer.

In design situations, many analyses has to be performed to study different geometries, loads and load combinations. It is beneficial if superposition techniques can be used since it significantly reduce the amount of analyses that needs to be performed. One approach that normally is used, is that the numerical analyses are performed as linear elastic and the sectional forces are extracted from the analysis. The sectional forces are then used to perform sectional design based on codes such as Eurocode 2, where nonlinearity (such as cracking) is assumed. The aim with design is not to perform as accurate calculations as possible, but instead to perform analyses with adequate conservatism to determine that the structure is safe, and satisfies design requirements.

Nonlinear analyses may be performed in design, but then these should be performed in a late stage of the design process, in order to verify the structural behaviour of the final design.

In evaluations of existing structures, it is more common to perform nonlinear analyses. The reason is that these types of analyses aims to determine the actual condition of the structure. As mentioned above, it is mainly nonlinearities caused by boundary/interactions and due to material nonlinearity that may have to be considered.

In analyses intended to determine strengthening solutions or structural rehabilitation, it is important that the nonlinear behaviour due to existing cracks etc. are considered in order to verify that the strengthening solution works as intended. Several different cases can be found in the literature where one rehabilitation measure was performed to prevent or reduce the influence from some effect, but after some years were found to be responsible for new problems, perhaps even in a different part of the structure. One example of this is insulation walls that were installed on several concrete buttress dams in Sweden, to prevent freeze-thaw damages on the frontal slabs. Unfortunately, most of these insulation walls were installed near the inspection gallery in the supporting buttress wall, which resulted in significantly increased stresses in these walls due to seasonal temperature variations, Malm and Ansell (2011).

2.4 DESIRED LEVEL OF SAFETY FOR DAMS

Different codes and guidelines have defined the desired level of safety, which differs depending on the limit state and depending on the potential failure modes, such as

- Global stability failure
  - × Overturning failure
  - × Sliding failure
- Material failure
  - × Compressive failure
  - × Tensile failure
  - × Shear failure

Some codes and guidelines base both the global stability failure and the material failure on global safety factors. In other codes, the partial coefficient method is used. In addition, some codes and guidelines also differentiates depending on the consequence class of the specific dam or parts within a dam structure.

Regardless of which safety concept is used, finite element analyses are a powerful tool to estimate the safety of the dam or the hydraulic structure, especially in cases with complicated geometries and load cases.

The safety concept is not an issue for linear analyses, since it is up to the FE-analysist to check the obtained results (through post-processing) according to the used design code or guideline. Thereby, in linear numerical analyses, the design values of loads can be defined directly and stress, sectional forces etc. are obtained from the FE-model and compared to the allowed value.

In nonlinear analyses, the safety concept is of significant importance. In Section 11.3, different safety concepts are presented. The safety concepts presented in this report are based on a partial coefficient method, such as those used in Eurocode or Model Code. In these analyses, it is defined that the material properties. These methods are based on ultimate limit analyses and the fact that reduction in strength will reduce the structural resistance.

For the global failure modes, this is not really the case. Material properties such as strength of concrete etc. are in general not of importance. In these analyses, the density of the materials and the coefficient of friction at the base of the dam are the two governing parameters for the resistance (and strength or rock bolts or tendons if these are considered).

In the current version of Swedish guideline RIDAS (2011), a value for the safety index is not defined. This is an on-going work and is described further in detail in Westberg and Johansson (2015).

According to BKR (2010), the β values are defined for the different safety classes as shown in Table 2-2.

Table 2-2 Safety index factors, annual failure probability and partial coefficient according to BKR (2010)

Safety class	Safety index β	pf	Partial coefficient. γ_n
Low	3.7	10^{-4}	1.0
Normal	4.3	10^{-5}	1.1
High	4.8	10^{-6}	1.2

2.5 GUIDELINES FOR NUMERICAL ANALYSES OF CONCRETE DAMS

There are many references in the literature that covers the aspects of numerical analyses of concrete dams and can be used as guidelines. Some of these were developed for a specific purpose, for instance time history analyses for seismic evaluation, while others are more general. In this section, a list of some of the guidelines found in the literature are presented.

The committee on Computational Aspects of Analysis and Design of Dams within the International Commission of Large Dams (ICOLD) have published several Bulletins regarding numerical analyses of dams:

- Bulletin 155 "Guidelines for use of numerical models in dam engineering", ICOLD (2013)
- Bulletin 122 "Computational procedures for dam engineering – Reliability and applicability", ICOLD (2001)
- Bulletin 94 "Computer software for dams – Validation, comments and proposals", ICOLD (1994)
- Bulletin 61 "Dam design criteria – Philosophy of choice", ICOLD (1988)
- Bulletin 53 "Static analysis of embankment dams", ICOLD (1986)
- Bulletin 30 – Finite elements methods in analysis and design of dams ICOLD (1987)

All ICOLD Bulletins can be downloaded from the ICOLD homepage: http://www.icold-cigb.org/GB/Publications/bulletin.asp [accessed 2016-01-04]

The US Army Corps of Engineers (USACE) have publish several guidelines regarding numerical analyses and design of concrete dams. Some examples are

- EM 1110-2-2104 "Strength Design for Reinforced-Concrete Hydraulic Structures", USACE (1992)
- EM1110-2-2200 "Gravity Dam Design", USACE (1995)
- EM1110-2-2201 "Arch Dam Design", USACE (1994)
- EM 1110-2-6050 "Response Spectra and Seismic Analysis for Concrete Hydraulic Structures", USACE (1999)
- EM 1110-2-6051 "Time-History Dynamic Analysis of Concrete Hydraulic Structures", USACE (2003)
- EM 1110-2-6053 "Earthquake Design and Evaluation of Concrete Hydraulic Structures", USACE (2007)

These can be downloaded from the following homepage: http://www.publications.usace.army.mil/USACEPublications/EngineerManuals.aspx [accessed 2016-01-04]

The Federal Energy Regulatory Commission (FERC) has published an engineering guideline for evaluation of hydropower projects. Especially the following chapters includes the topic of numerical analyses of concrete dams

- Chapter 3 "Gravity Dams", FERC (2002)
- Chapter 10 "Other Dams", FERC (1997)
- Chapter 11 "Arch Dams" , FERC (1999)

The engineering guideline is available from the following homepage: https://www.ferc.gov/industries/hydropower/safety/guidelines/eng-guide.asp [accessed 2016-01-04]

Another example of a guideline have been developed by USBR (2006) focused on nonlinear structural analyses of concrete dams.

- USBR (2006) "State-of-Practice for the Nonlinear Analysis of Concrete Dams at the Bureau of Reclamation".

This guideline can be downloaded from the following homepage http://hdl.handle.net/2027/mdp.39015064772372 [accessed 2016-01-04]

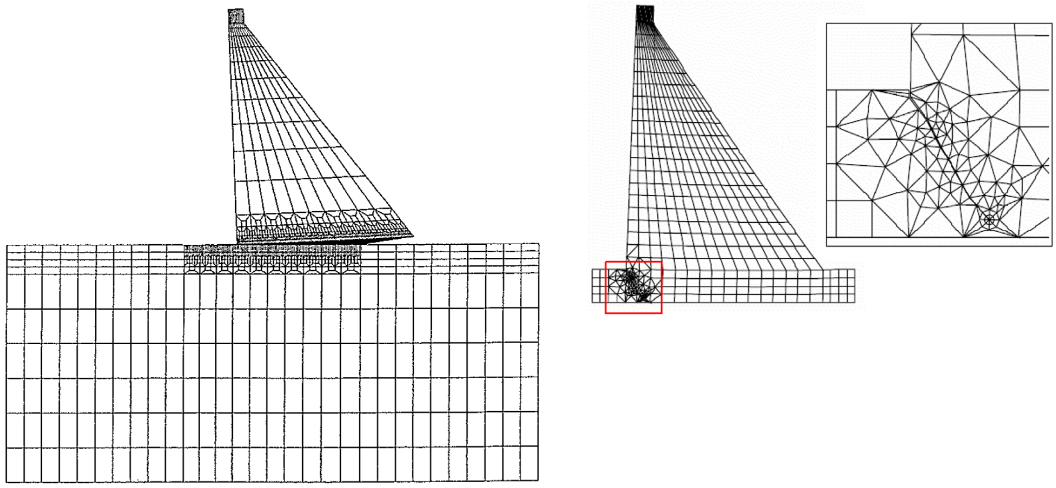
3 Geometrical model

3.1 GEOMETRICAL MODEL OF THE CONCRETE DAM

Due to the geometry of concrete dams, three different types of analyses are mainly performed as

- 2D plane strain or stress
- 3D solid models
- 3D shell models

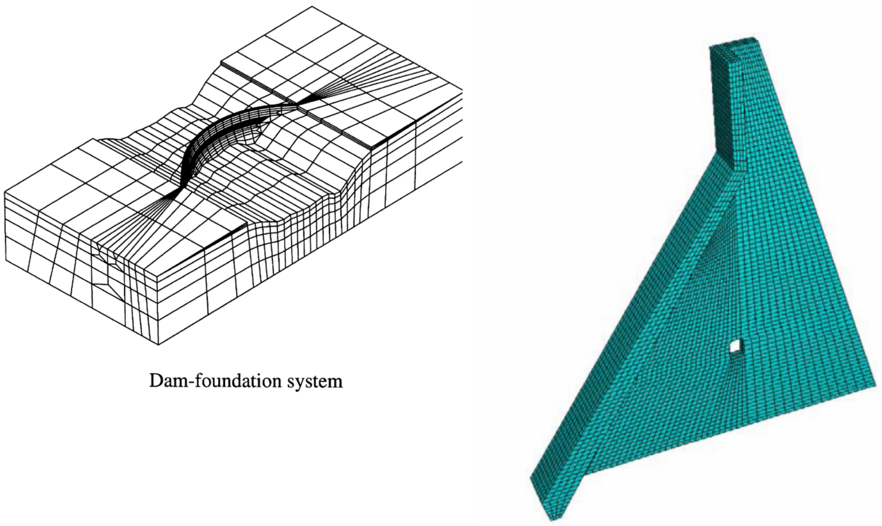
Dams with a continuous geometry such as gravity dams (including RCC dams) consist in general of a constant cross section with expansion joints defined typically 12 – 15 m apart. For these types of dams, it is often preferred to perform 2D analyses based on plane strain theory. If the analyses considers nonlinear material behaviour, i.e. cracking, due to shrinkage it is often preferred to perform the 2D analyses based on plane stress theory in order to prevent out of plane cracking in the model. Two examples of concrete gravity dams that have been analysed with 2D models are shown in Figure 3-1.



a) b)

Figure 3-1 Examples of 2D analyses, a) sliding of a gravity dam (Linsbauer and Bhattacharjee, 1999), b) gravity dam with a fracture in the rock at the upstream toe (Ruggeri, 2004)

For cases where there is a variation in cross-section along the dam line and/or if the dam transfers load in the lateral direction, 3D analyses with solid elements are mainly used. These types of dams are typically arch dams which carries a significant part of the load through arch action (depends on the design and load condition of the dam) or other types of dams such as concrete buttress dams. Concrete buttress dams may also be analysed in 2D, but then an important factor such as lateral bending of the front-plate will be disregarded. Some examples of cases with different types of dams that have been studied with means of 3D solid elements are shown in Figure 3-2 and Figure 3-3.



a) b)

Figure 3-2 Examples of 3D analyses, a) Seismic analysis of an arch dam (Malla and Wieland, 1999), b) Thermal stress distribution due to seasonal variation in a concrete buttress dams (Björnström et al., 2006).

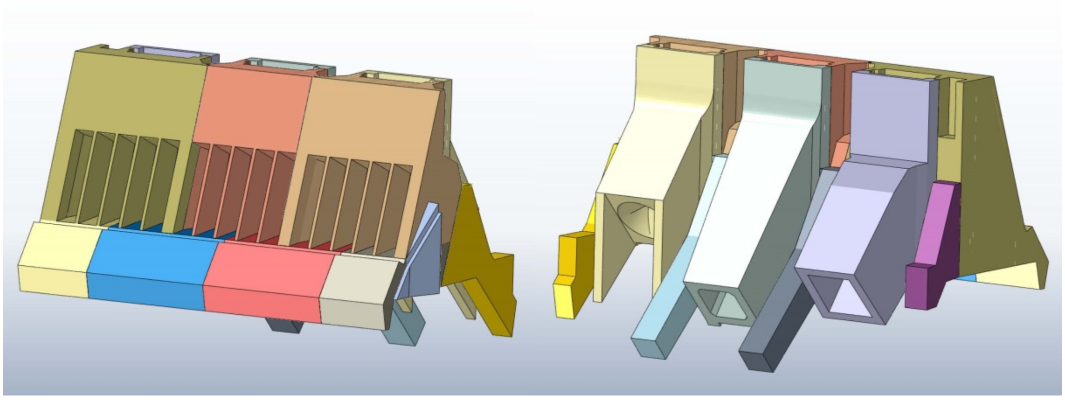


Figure 3-3 Examples of a model used in 3D analyses, of Krångfors intake structure that have been redesigned/upgraded in three stages, up- and downstream view, from Johansson et al (2015).

The more slender type of dams, such as ambursen and concrete buttress dams may also be defined with shell elements instead of 3D solid elements. The advantage with using shell elements are

- Faster analyses
- Sectional forces can be obtained directly

The disadvantage is that it can be difficult to constrain shell elements to each other in complicated corners etc. so that the stiffness is correct. As default, shell elements are representing the mid plane of for instance a slab. In order to constrain two shell elements in a corner it is important that the shells are defined with an offset so that the actual stiffness is represented. This is for instance illustrated in Figure 3-4. In addition, using shell elements to perform analyses of cracking aims to analyse the effect on a global scale, if more detailed information regarding crack propagation in the thickness direction etc., are desired then solid elements are required. An example of crack propagation analyses based on a 3D shell element model of a buttress dam monolith is shown in Figure 3-5

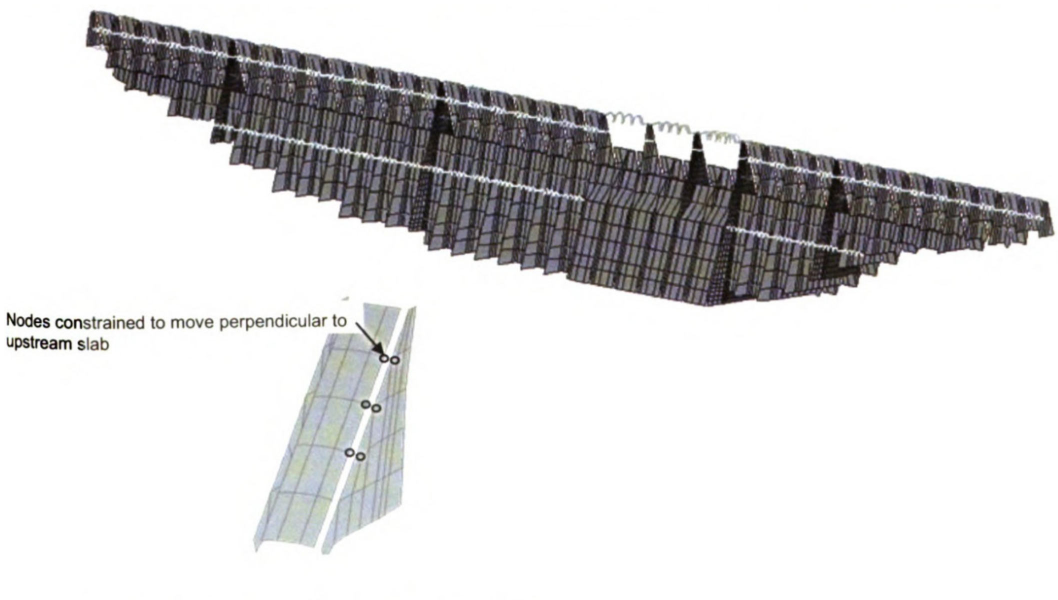


Figure 3-4 Examples of Stony Gorge Dam of Ambursen type that been analysed with shell elements. From USBR (2006).

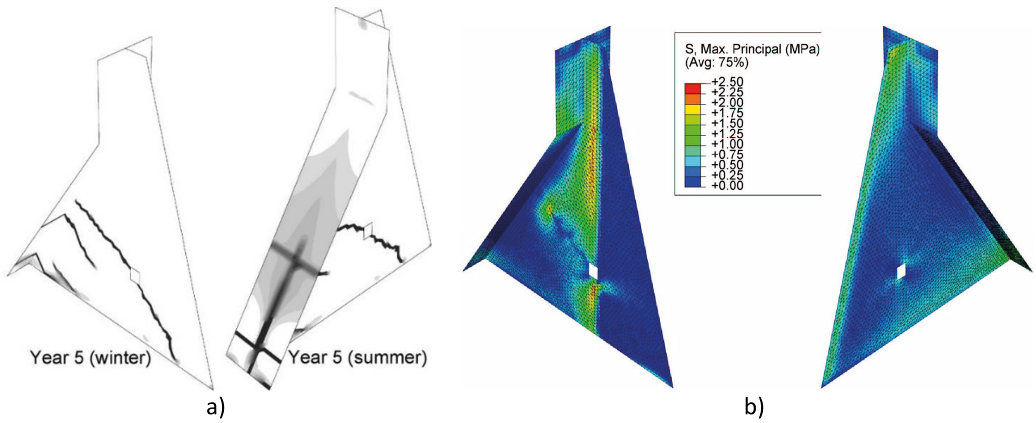


Figure 3-5 Examples of one monolith of Storfinnforsen concrete buttress dam that been analysed with shell elements, a) cracking due to seasonal temperature variation (Malm and Ansell, 2011), b) Induced stresses due to placement of an insulating wall; near inspection walkway, right: downstream (Isander et al 2013).

3.2 CONSIDERATIONS FOR DEFINING THE GEOMETRICAL MODEL

One aspect that might influence the accuracy of the numerical model is regarding how the large part of the actual dam that is included in the analyses. If too small region is included, the boundaries may affect the result.

The size of the model depends on the purpose with the analyses, which aspects the models needs to describe. The model must be able to capture the mechanical mode of action of the parts that should be analysed. In many cases, it may be sufficient to study one monolith, while in some cases the whole dam (especially for arch dams) may have to be studied. In each project, an engineering judgement has to be made regarding the geometric size of the model and that the level of detail is sufficient to capture the aspects of interest.

3.2.1 Limitation of the model due to contraction joints

For instance, when analysing long dams consisting of several monoliths, spillways, etc. which are divided by contraction joints, it is most often sufficient to limit the model to only include the parts within the contraction joints. This is a valid assumption if the dilatation/contraction joint works as intended, i.e. where no forces are transmitted over these joints. These joints are built with a water-stop, often consisting of bitumen, PVC or steel. However, in old structures, the size of the dilatation joint may have reduced over time or the joint has become more a more rigid which could lead to a case where forces are transmitted across the joints.

3.2.2 Considering the rock mass

Another example is regarding how much of the rock that should be included in the numerical analysis. This is discussed further in detail in Section 6.1. Three different approaches are mentioned

- Define boundary condition at the base of the dam (i.e. no rock is included). – not a common approach due to its limitations, see Section 6.1.1
- Use springs or equivalent special purpose elements to represent the spring stiffness – see Section 6.1.2
- Include large part of the surrounding rock mass – See Section 6.1.3

If the rock mass is defined with the same size of elements as the dam, then this would result in huge models where the majority of the degrees of freedom are within the rock mass, i.e. in a part of the model where we are not really interested of the results. Thereby, a biased seeding of edges is recommended be used in the model where larger elements can be used in the rock further away from the dam. In addition, most software can with reasonable good accuracy constrain two surfaces with large differences in discretization. Thereby it possible to mesh the rock foundation with coarser mesh at the interface with the concrete dam. One recommendation in these cases is to imprint the outline of the dam on the foundation and thereby the ensure that the nodes of the outline of the dam have a relatively short distance to the closest nodes on the foundation.

3.2.3 Reduce the model based on symmetry

To reduce the size of the model, it is often possible to use the symmetry that is given. Concrete monoliths are in general symmetric along its centreline and hence in many cases it is sufficient to only model one of the symmetric halves and define a symmetry boundary condition on the centre line. This will however, not be possible in those cases where the load may act unsymmetrically in the lateral direction over the monolith.

It is not recommended to use symmetry planes in models where the nonlinear material behaviour is considered, i.e. concrete cracking. The reason is that in a model with a symmetry where cracks occur near the symmetry plane results in an overestimation of the softening of the structure.

3.3 USING THIRD PARTY SOFTWARE TO GENERATE THE GEOMETRY

Most FE software's have good capabilities of defining complex geometries in their pre-processor. However, today it is common that the geometry is defined in 3D-CAD software's and imported into the FE-software. Most FE-codes can import general file types such as .sat, .iges, etc.

One potential problem that can arise from this procedure is that the CAD model is that there are different aims with 3D CAD models and FE models and in many cases, it is necessary to simplify the CAD geometry model in order to be able to define a suitable FE model. One other thing is that the FE code uses high tolerances for intersecting lines and even if two nodes that should coincide, differ slightly the FE code will not find the intersection unless certain importing tools are available in the FE software. A third thing that might occur is that, in the CAD-model it does not really matter if the geometry is defined with a few faces or defined as a solid. In a FE model, these faces will be interpreted as shell elements instead of solid elements. In conclusion, it can be said that the FE-model requires higher geometric accuracy than a normal CAD model but lower level of detail.

It is therefore important that the FE-engineer is part in the process of developing the CAD geometry and it might require some iterations between the CAD-engineer and the FE-engineer in order to develop a suitable geometry.

There are also many software's available that are specialized in developing a mesh for the structure. Most FE codes have their own mesh module in their pre-processor and some are rather sophisticated while others leave much to desire. In the same way as for the CAD software's, the mesh generated by specialized mesh-software's may be imported into the FE software's.

4 Element discretization (Mesh)

In this chapter, a very brief description of different element types is given. This section is primarily directed to beginners within FE analyses.

There are several different types of elements in a normal element library in a commercial finite element code. In this section, the most common types of elements used in mechanical (structural) analyses are described briefly. Below, the most common types of elements and their degrees of freedom are given:

- Truss elements (only translations at the nodes, i.e. no bending)
- Beam elements (translations and rotations at the nodes)
- Shell elements (translations and rotations at the nodes)
- Solid elements (only translations at the nodes, i.e. no bending)
- 2D plane stress / plane strain
- 2D axisymmetric

In addition to these types of elements, there are other element classes, such as membrane, springs and dashpots that are commonly used in structural analyses, as seen in Figure 4-1. In some software's, other types of elements are available such as plate elements, which basically can be described as simplified version of a shell element intended for out of plane forces only.

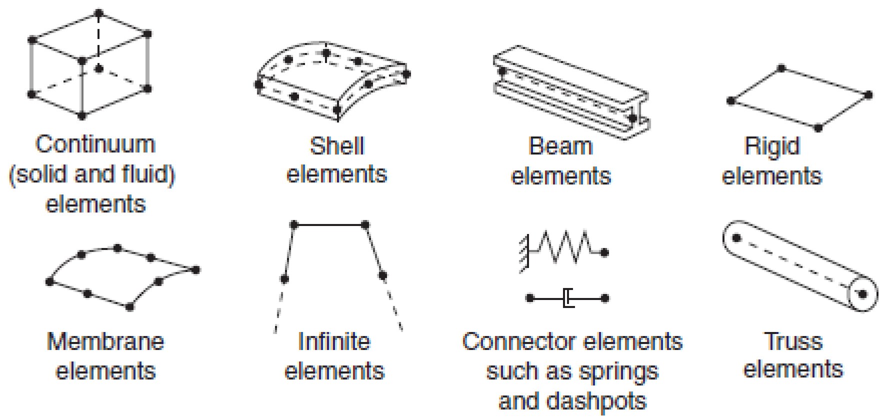


Figure 4-1: Common element classes in structural analyses, from Dassault (2014).

In most cases, it is possible to simplify the structure that is going to be analysed and yet still capture its global behaviour. Often this step is one of the most difficult tasks for an inexperienced engineer. When this simplification of the real structure into a numerical model is made, it is important to know the limitations of the element classes.

As a simple example, it can be mentioned that solid elements and truss elements only have degrees of freedom for translations (displacements) while shell and beam elements also have rotational degrees of freedom. Thereby, by modelling a structural part (a reinforcement for instance) with truss elements results in that there is no bending stiffness included (i.e. dowel effects etc. are neglected).

As mentioned earlier in Section 3.1, the concrete dams are often defined with solid elements (in 2D or 3D). Despite this, there are many other types of elements that may be required in a numerical model. These can for instance be secondary structural parts or elements only intended as modelling aid. Below, a few examples of types of objects/structural parts that might be included in the model.

- Point mass element → mass from secondary structures, mass of water in Westergaard approach etc.
- Truss elements → truss structures, reinforcement bars, cement grouted tendons, etc.
- Beam elements → beam or frame structures, reinforcement bars, rock anchors, tendons, etc.
- Membrane elements → modelling aid (for instance to embed reinforcement layers within a solid) etc.
- Shell elements → thin slab buttress dams, intake gates, etc.
- Solid elements → almost everything such as dams (gravity, arch, buttress dams), rock, etc.

It is often the sectional forces (normal force, shear force and moment) that are of interest in design of structures. These outputs are however only obtained from beam and shell elements. However, in solid elements these outputs are not generally provided, instead stresses are given. In order to obtain sectional forces, the stresses have to be integrated over the cross-section. An alternative approach to obtain sectional forces from FE models based on solid elements is presented in Section 12.5.

Within each element class, there are also different types of elements that may be used in different applications. For instance, there are two main different beam elements

- Euler-Bernoulli (neglects shear deformations)
- Timoshenko (includes shear deformations)

The difference between these two elements is that shear deformations are not considered in Euler-Bernoulli, i.e. plane sections remain plane. This assumption is common in design calculation methods and is valid for slender cross-sections. For cases with more massive cross-sections, the shear deformations will significantly influence the behaviour.

In the same way, there are the following two different types of shell elements whether the shear deformations or not

- Kirchhoff (neglects shear deformations)
- Mindlin (includes shear deformations)

In Kirchhoff elements, the shear deformations are neglected and hence this assumption is only valid for slender (thin) shells, while Mindlin elements are intended for shells with a larger thickness.

4.1.1 Element shape functions

The method used in finite element theory is based on that a certain number of discrete values (nodes) describes the full behaviour of a structure and between these nodes, the solution is interpolated. This interpolation is made with shape functions for the elements. Between the nodes, linear or quadric interpolation may be used, as illustrated in Figure 4-2. The accuracy of the solution is to large extent dependent on this interpolation. In many cases, output are needed which is integrated from the interpolated values and its derived functions.

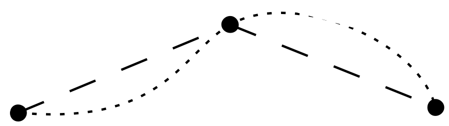


Figure 4-2: Linear or quadric interpolation between nodes, from Malm (2015).

In Figure 4-3, two types of 2D plane stress or plain strain elements are illustrated. The left element is a four-noded element, where nodes are defined in each corners. Along each edge, only a linear interpolation may be used since only two corner nodes can be used for interpolation. In the right figure, an eight-noded element with nodes defined in each corner, and one additional node along each edge, midway between the two corner nodes is shown. For this type of element, a quadric interpolation may be used since three nodes are available to provide the interpolation.

One important fact when using elements with linear shape functions is that it can only make a linear estimation of the structural geometry, regarding both its initial undeformed geometry but also its deformed geometry after subjected to loading. The four-noded elements are thereby quite poor to describe bending. A quadric element may be bent, in order to provide a quadric adjustment to the structural geometry. Regarding calculation cost, the higher order elements, such as the eight-noded element, is of course more expensive and will quickly increase the total degrees of freedom of the model with twice as many degrees of freedom for each element. However, since higher order elements are more accurate especially for describing bending, they often allow for coarser meshes and thus fewer elements, as later shown in Section 4.1.2. This can sometimes compensate for the increased computational cost per element.

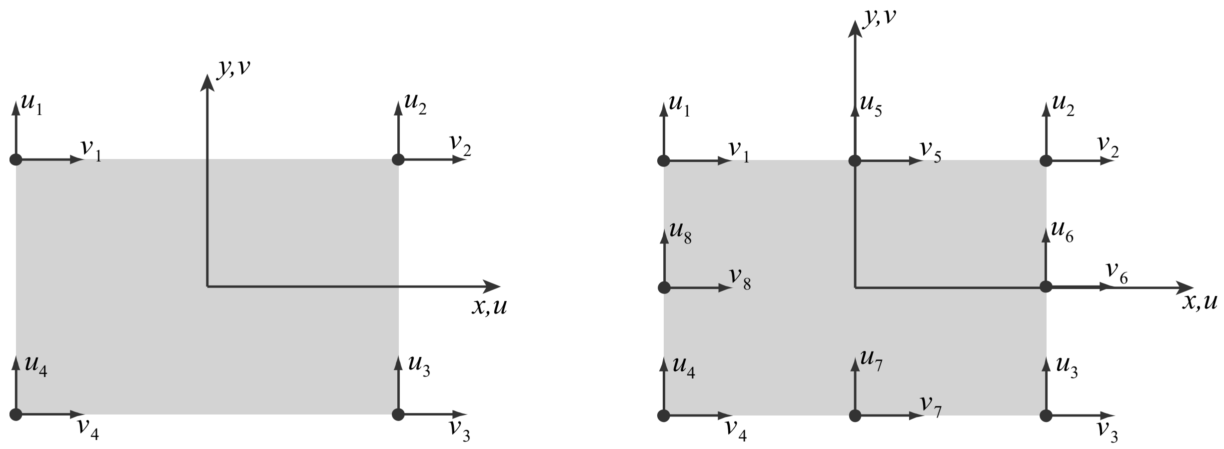


Figure 4-3: Element types for 2D plane stress elements. left) four-noded elements, and right) eight noded elements. From Malm (2015).

4.1.2 Discretization

As a general rule of thumb, it can be said that the accuracy of the FE analysis will increase with more detailed discretization of the structure, i.e. by using more elements or higher order elements. There is thereby, two ways to increase the accuracy of the numerical analysis, increase the number of elements or replace lower order elements (with linear shape functions) with elements of higher order (quadric shape functions). This is however, only valid if the discretization is made in suitable and accurate manner. Important facts when defining the mesh of the structure is to try to avoid using distorted elements (for instance too skew elements) and try to use elements with an aspect ratio close to one, i.e. height and width of the element is close to each other as illustrated in. This will be further illustrated in an example below.

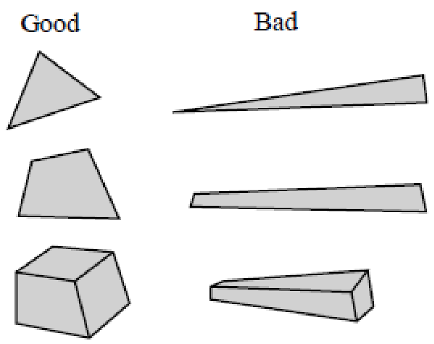


Figure 4-4 Elements with good and bad aspect ratios, respectively. From Felippa (2004)

A few examples are illustrated in Figure 4-5, where a cantilever beam subjected to a point load on its free edge have been analysed with 2D plane stress elements. In the figure, the result from six analyses with different discretization's of the structure is shown. The figure also shows an estimation of the error obtained from the numerical analysis regarding estimating the displacement of the cantilever beam. In all analyses, except one – case c), four-noded elements are used.

This comparison illustrates a few important facts,

- Distorted elements give lower accuracy, compare case a) and d)
- The accuracy is normally increased by introducing more elements, compare a), b) and e).
    - × However, this does not applies to cases were this leads to poor aspect ratio of the elements, compare case a) and f).
- The aspect ratio should be close to 1, i.e. height / width ≈ 1
- Higher order elements (eight noded instead of four-noded) results in higher accuracy.

One other important conclusion is that with too few elements, i.e. to large elements, the structural stiffness is over-estimated and hence the deflection of the structure is under-estimated.

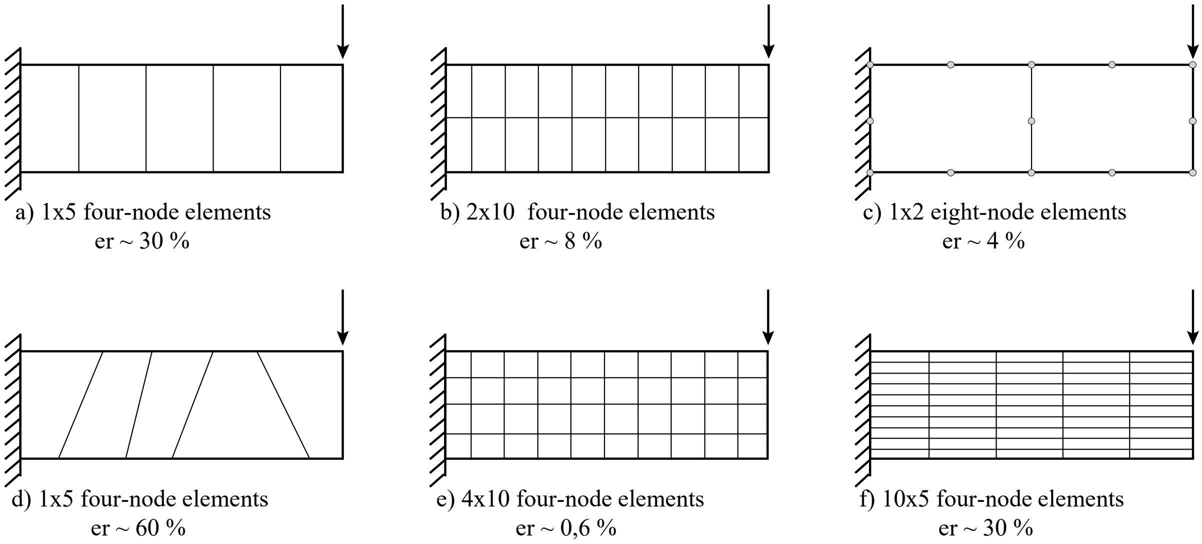


Figure 4-5: Example to illustrate the difference in error (er) for different discretization's. a) 1x5 4-noded elements, b) 2x10 4-noded elements, c) 1x2 8-noded elements, d) 1x5 4-noded elements, e) 4x10 4-noded elements, f) 10x5 4-noded elements. From Eriksson (2002).

When using finite element analyses, one should always perform convergence test analyses, see Section 4.2. This means that two analyses are performed with different discretization's, for instance with different element sizes, or linear compared to quadric elements. If the results from these two analyses are identical (as far as possible, except possible small round off errors), the larger element size is considered sufficient to be used for further analyses.

Based on the type of analysis that is being performed, different resolution of the model may be required. For instance, in cases where the nonlinear behaviour of concrete (cracking and crushing) is analysed, significantly smaller elements may be required than in a corresponding linear analysis. Malm (2006) presents that it is generally, better to use many (i.e. small elements) but fairly simple elements rather than using fewer higher order elements.

4.1.3 Element integration points

Displacements and rotations are calculated in the nodes of the element, and only for the active degrees of freedom since the inactive ones are per definition equal to zero. By using the calculated displacements and rotations, the linear static equation may be used again to calculate the reaction forces/moments for inactive (i.e. constrained) degrees of freedom.

Based on the calculated displacements at the nodes, the strain in the element can be calculated in the element integration points (also called Gauss points). Based on the strain, the corresponding stresses may be calculated in the element integration points.

The integration points in 4-noded elements and 8-noded elements are shown in Figure 4-6. For both the 4-noded and the 8-noded element, two set-ups of integration points may be available, corresponding to either full integration or reduced integration.

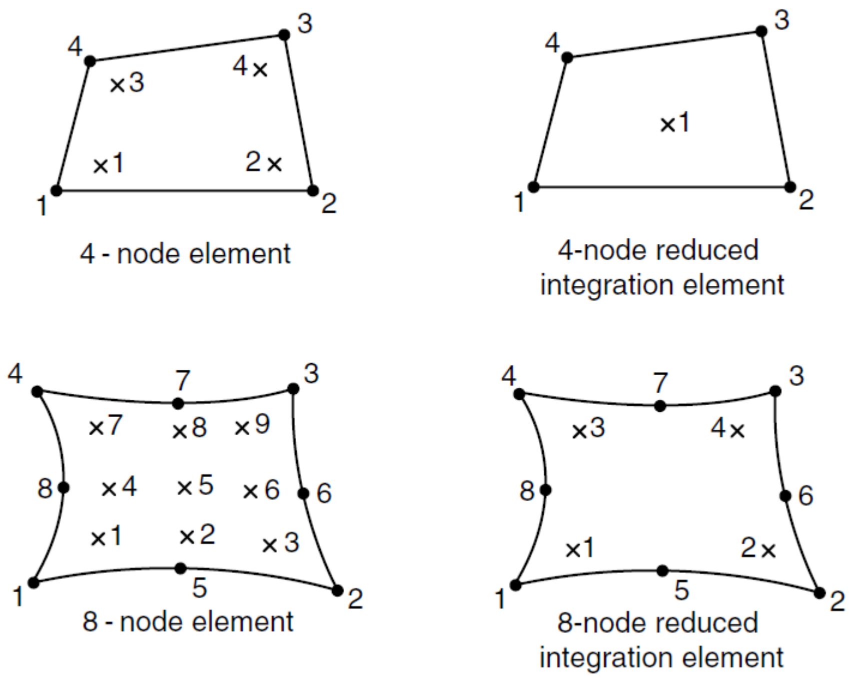


Figure 4-6: Integration points in 4-noded and 8-noded elements, from Dassault (2014).

The advantage of using elements with reduced integration is of course that analysis requires less cpu-time. However, there are risks and disadvantages when using reduced integration elements. One significant risk is that hourglass modes occurs in the elements. The main risk with hourglass is that these deformations of the elements results in zero energy modes, which underestimates the stiffness and stresses of the structure.

For instance if only one integration point is used in a 4-noded element, the strain/stress in the element will be zero if the element is subjected to deformations that causes one side to shrink and the opposite side to expand with the same value, i.e. so that the element is distorted like a trapezoid. This is illustrated in the left figure of Figure 4-7, where the strain in the centre of the element will be zero for this deformation. The same phenomenon where some deformations may cause zero stress/strain in the integration points can also occur in an 8-noded element. For an 8-noded element, this deformation mode looks like the right figure in Figure 4-7. This type of deformation modes that does not cause any strain or stress is called hourglass modes. The name hourglass modes, comes from the shape of the deformation mode of the 8-noded element where it is shaped like an hourglass. The hourglass mode shape for the 4-noded element would normally occur in all cases where we model a beam, which is subjected to a moment. However, in many FE programs there are methods to suppress these hourglass modes and thereby reduce their tendency to occur in calculations based on elements with reduced integration.

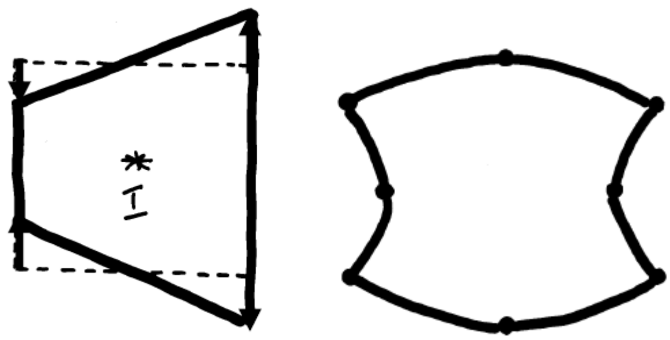


Figure 4-7: Hourglass modes for four noded and eight noded elements. From Eriksson (2002).

4.2 CHOOSING SUITABLE ELEMENT SIZE

Depending on the type of structure, type of elements used, type of analysis and the desired level of detail (global or local effects) different types of element discretizations may be required.

One should always perform convergence analysis tests depending on mesh size (or element type).

In a linear analysis, the structural stiffness will be overestimated if an insufficient discretization is used. This means that in a buckling analysis, the load carrying capacity will be overestimated.

The following mesh control of the results regarding discretization should be performed in all projects according to Ekström et al. (2016).

- Increased discretization (smaller elements or using elements of higher order) should result in a marginal difference in result.
  - × Note that deformations and loads will converge faster than strain or stresses. (deformations and loads are calculated at the nodes while strain and stresses are calculated within the element)
  - × The convergence test should be conducted so that the same problem is analysed with different discretization’s, i.e. models with element size. By comparing the results from these different models only marginal

difference in results should be obtained when going from one coarser mesh to a more detailed mesh. If the FE-analysis is intended to perform a stability evaluation of a concrete dam, one could for instance compare the results for different mesh refinements and if the difference in crest displacement is typically, less than 1 % between two analyses it can be considered accurate enough.

In addition to mesh convergence controls, other types of verifications and validations are required, as later described in Chapter 10.

The size of the elements in a model should be chosen so that relatively distributed results obtained, i.e. there should not be any significant spikes in the results from one element to another.

- Since stresses requires higher level of discretization (i.e. does not converge as fast as displacements), the FE-engineer should study the stress distributions within the structure to make sure that the element discretization is sufficient.
- Most software's present the contour plots with averaging and therefore the FE-engineer should compare the result with and without averaging to ensure the element discretization.

In analyses based on continuum elements, point loads and point boundary conditions will introduce unreasonable results with significant stress/strain gradients. For a linear elastic analysis, the stresses at the element closest to these point loads or boundary conditions will reach infinity for an infinitesimal element size, as illustrated in Figure 4-8. This is the reason why these types of loads and boundary conditions should be avoided as far as possible, as seen in Chapter 7.

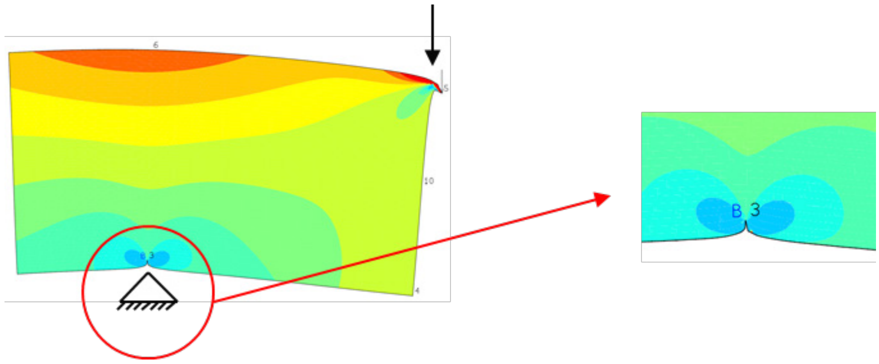


Figure 4-8 Singularity at point boundary conditions and point loads, from Pacoste (2001).

Different cases where significant mesh sensitivity is likely to occur in FE models are illustrated in Figure 4-9. In these areas, locally refined mesh is required, i.e. smaller mesh size is required compared to the remaining part of the models, due to large stress/strain gradients. This occurs as mentioned previously near concentrated loads, but also near abrupt changes in thickness, material properties etc. In the figure, entrant corners refers to a region where the principal stress trajectories meet and "bunch up", (Felippa 2004).

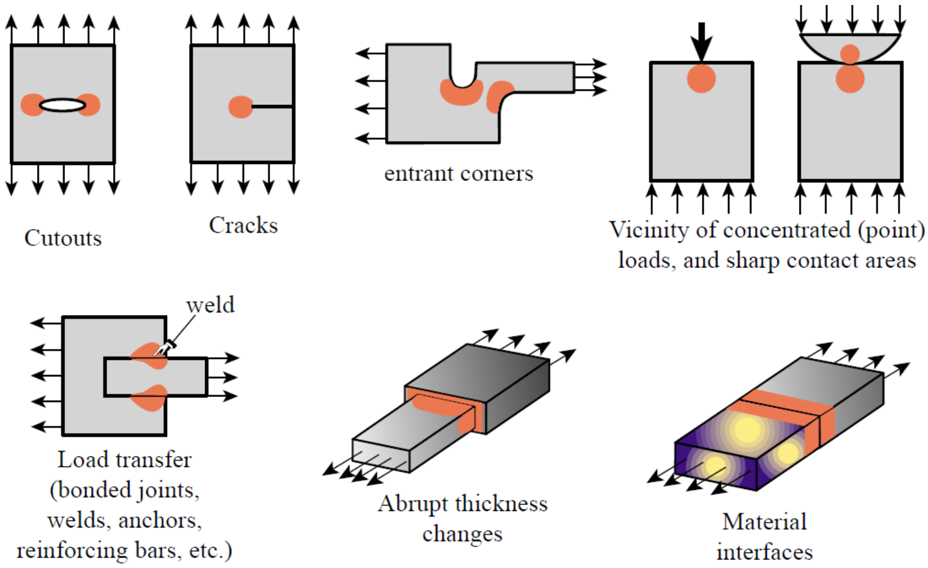


Figure 4-9 Typical areas in a model where locally refined mesh is required. From Felippa (2004).

4.2.1 Linear elastic analyses

In linear elastic analyses, the requirement on a dense mesh is not as strict as in the case with nonlinear analyses.

In linear elastic analyses, it may be sufficient with only two elements defined in the thickness direction, if high-order elements (based on quadric shape functions) are used, see Figure 4-5.

If lower order elements with linear shape functions are used, it may be required to use five elements in the thickness direction, since these elements gives a poor description of bending, as seen in Figure 4-5.

4.2.2 Nonlinear analysis of concrete cracking

In nonlinear analyses where cracking of concrete is described with continuum approach, much smaller element lengths are required compared to corresponding static analysis. The reason for this is that all the stored strain energy within the element will be released due to cracking and this amount of energy must be absorbed by the surrounding elements to prevent that a snap-back behaviour occurs. According to the Dutch guideline for nonlinear analyses (Rijkswaterstaat, 2012), the maximum allowed element length for a case with an exponential unloading stress-displacement curve is

$$L_{max} < \frac{E \cdot G_f}{{f_t}^2}$$

where,

E is the elastic modulus [Pa]

G_f is the fracture energy [Nm/m²]

f_t is the tensile strength [Pa]

As an example, it can be mentioned that for a conventional C30/37 concrete with material properties according to EN 1992-1-1 (Eurocode 2, 2008), i.e. mean values $E = 33$ GPa and $f_t = 2.93$ MPa in addition to the fracture energy based on fib Model Code (2010) equal to $G_f = 140\ \mathrm{Nm/m^2}$ requires an element length that is less than 0.5 m.

Recommendation regarding element lengths are given in Table 4-1 for solid and shell elements in numerical analyses of cracking. It should be noted that these recommendations of the maximum element length is not a guarantee that convergence is obtained. Smaller element lengths than this may be required in the analyses than given in these examples. In the table, l, h and b corresponds to the length, height and width of the beam/slab respectively.

Table 4-1 Recommended maximum element length in nonlinear analyses, from Rijkswaterstaat (2012).

Beam	$L_{\max}$	Slab	$L_{\max}$
2D model	$\min\left(\frac{l}{50},\frac{h}{5}\right)$	2D model	$\min\left(\frac{l}{50},\frac{b}{50}\right)$
3D model	$\min\left(\frac{l}{50},\frac{h}{5},\frac{b}{5}\right)$	3D model	$\min\left(\frac{l}{50},\frac{b}{50},\frac{h}{5}\right)$

4.2.3 Seismic analyses

In seismic analyses, it is also important that the element size is small enough in order for it to be able to describe the response of the full frequency content of interest.

As a rule of thumb, the following equation can be used to define the largest element size that can be used in a dynamic analysis depending on the highest frequency of interest

$$L_{max} = \frac{c}{n_{min} \cdot f_{max}}$$

where,

c is the shear wave velocity of the material [m/s]

n_{min} is the minimum number of points that is required to approximate a sine wave

f_{max} is the highest fundamental frequency of interest (for the structure) [Hz]

The, shear wave velocity of the material can be calculated as follows

$$c = \sqrt{\frac{E}{2(1+\nu)\rho}}$$

where,

E is the elastic modulus [Pa]

ν is the poisons ratio

ρ is the density [kg/m^3]

According to USBR (2006), there should be no fewer than 10 elements per wavelength, i.e. $n_{min} = 10$. Thereby, in seismic analyses of a concrete dam a minimum element length of 6 m is in general required. This is based on a calculation where the shear wave velocity is about 3000 m/s, the highest frequency of interest is 50 Hz and that 10 points are sufficient to describe the wave.

5 Material behaviour

In this chapter, recommendations regarding the choice of material properties and the material behaviour of concrete and steel (reinforcement and tendons) are given. In addition, this chapter includes a brief introduction to material nonlinearity (cracking, crushing etc.) and how this could be considered in numerical analyses with suggestions on choice of material properties etc.

There are several more aspects causing nonlinear behaviour of concrete or steel, caused by long-term effects (creep, shrinkage), degradation (leaching, ASR, frost, corrosion etc.), fire, etc. Shrinkage and creep are described in 5.1.4 and 5.1.5 respectively. The other aspects, are not covered in this report.

All material properties should be based on measurements performed on the type of concrete at the specific site as far as possible. However, in many cases, measured data may not be available or only a few material properties have been measured. In those cases, the material properties should be based on valid design guidelines or standards/codes. However, in numerical analyses, some material properties may be required as input in a specific material model and these are normally not given by these guidelines or codes.

Therefore, in the following section recommendation regarding material properties and material behaviour is given for concrete and steel based on Eurocode 2 (2008) as far as possible. In several cases, additional material properties have been obtained from other sources, such as fib Model Code (2010).

When performing numerical analyses in order to assess/evaluate the condition, load capacity etc. of a structure, "best-estimate" material properties should always be used. This means that if codes etc. are used to estimate the material properties, then the material properties corresponding to the mean values should be used in general.

5.1 CONCRETE

In this section, the concrete material properties are presented for concrete based on Eurocode as far as possible. In cases where some material properties are lacking in Eurocode, then Model Code has been used as reference instead.

It should be mentioned that ICOLD bulletin 145 is on the topic of physical properties of hardened conventional concrete in dams ICOLD, ICOLD (2016). This report includes background information on the material behaviour and testing methods etc.

5.1.1 Linear elastic material properties

In order to describe the linear behaviour of concrete in mechanical analyses, typically only two or three material properties are needed. These are also rather well defined for concrete; elastic modulus, poisons ratio and density. The density is only needed in static analyses if gravity loads are considered but it is always needed in dynamic analyses since the mass will result in inertia forces.

The mean value of the of the elastic modulus of concrete 28 days after casting, is given in Table 3.1 in Eurocode 2 and varies between 27 - 44 GPa for concrete grades C12/15 to C90/105. The elastic modulus given in Eurocode 2 is based on the following equation

$$E_{cm} = 22\left(\frac{f_{cm}}{10}\right)^{0.3}$$

where

E_{cm} is mean elastic modulus of concrete [GPa]

f_{cm} is mean cylinder compressive strength of concrete [MPa]

The elastic modulus is not constant over time. It will increase during hardening (which is further described in Section 5.1.3) and creep is often considered as a reduction of the elastic modulus (as further describe in Section 5.1.4).

The density of normal unreinforced concrete varies typically between 2000 – 2600 kg/m^3 according to fib Model Code (2010). According to Eurocode 1 (2011), the weight should be assumed equal to 24 kN/m^3 and with normal amount of reinforcement (bars or tendons) or for cases with young unhardened concrete as 25 kN/m^3. In stability analyses of concrete dams, the gravity load is beneficial, therefore, a lower value of the density is given in RIDAS (2011) corresponding to a weight of 23 kN/m^3.

The poisons ratio is normally considered to be within an interval of 0.1 and 0.2 for hardened concrete. During hardening of concrete, measurements, Byfors (1980), have shown that the poisons ratio varies significantly. In Eurocode 2, it is given that the poisons ratio may be assumed equal to

$$\nu = \begin{cases} 0.2 \text{ for uncracked concrete} \\ 0 \text{ for cracked concrete} \end{cases}$$

If thermal expansion/contraction is considered in the FE-analysis, then a thermal expansion coefficient (α) is required. The thermal expansion of concrete may vary depending on type of constituents and during hardening of concrete. The thermal expansion is in general between $0.74 \cdot 10^{-5} \leq \alpha \leq 1.3 \cdot 10^{-5}$ [K^{-1}] for concrete based on Portland cement according to FHWA (2011). According to Eurocode 2, the thermal expansion may be assumed equal to

$$\alpha = 1.0 \cdot 10^{-5} \text{ [K}^{-1}\text{]}$$

One difficulty when calculating thermal stresses, is to determine the reference temperature T_{ref}. This reference temperature corresponds to the temperature when it is assumed to be stress-free. This temperature is often also denoted as the closing temperature, i.e. the temperature when the final segment is casted that causes the structure to be restrained. According to Eurocode 1-5 (2003), the reference temperature can be considered to be +10 °C, unless better estimation of this temperature is known.

5.1.2 Nonlinear material properties for cracking or crushing

Depending on which theory the material model for nonlinear behaviour of concrete is based on, different type of material properties may be specified and in addition, each specific model may also require some modelling parameters which may not be directly related to a physical property of the concrete and therefore has to be calibrated.

Common for all material models is that the uniaxial curves in tension and compression respectively, have to be defined. All material models also include some curve for the biaxial behaviour of concrete, where the FE-analysist has to specify how the strength develops during biaxial stress states. In addition, the shear resistance of a crack have to be defined almost regardless of which material model is used. The tri-axial response is also commonly required as input. Thereafter, the number of properties that has to be defined depends on the complexity of the material model. In general, the more complex the material model is, the more properties and modelling parameters have to be defined. One difficulty is however, that many of these material parameters are not conventionally tested in standardized tests and may therefore have to be assumed or estimated.

In the following section, recommendations regarding material properties that are required in the most common material models are given. For further description of the nonlinear behaviour of concrete, the following references may be studied Malm (2006), Malm (2009), Mang et al. (2003) etc.

Uniaxial tensile behaviour

The 28 day tensile strength can be obtained from Table 3.1 in Eurocode 2 and the mean value varies between 1.6 - 5 MPa for concrete grades C12/15 to C90/105. The tensile strength may be determined based on the following equation according to Eurocode 2

$$f_{ctm} = \begin{cases} 0.30{f_{ck}}^{2/3} \text{ for } \leq C50/60 \\ 2.12 \cdot \ln\left(1 + \frac{f_{cm}}{10}\right) \text{ for >C50/60} \end{cases}$$

where

f_{ctm} is mean tensile strength [MPa]

f_{ck} is the characteristic cylinder compressive strength, ($f_{ck} = f_{cm} - 8$ [MPa]) [MPa]

In order to fully define the uniaxial tensile behaviour of concrete, fracture energy and the shape of the unloading curve are required.

These are not given in Eurocode, instead the fracture energy may be determined based on fib Model Code (2010). In absent of experimental data, Model Code 2010 gives the following expression that can be used to determine the fracture energy (mode I) for normal weight concrete

$$G_f = 73 \cdot {f_{cm}}^{0.18}$$

where

f_{cm} is mean cylinder compressive strength of the concrete [MPa]

It should be noted, that the fracture energy is not dependent on the aggregate size in the equation above. The fracture energy in normal weight concrete depends primarily on the water/cement ratio, the maximum aggregate size and the age of concrete. In dams and other hydropower structures, large aggregates are normally used which thereby allows for quite high fracture energy.

The last thing that is required to define the uniaxial behaviour of concrete is to define the shape of the unloading curve, i.e. the crack opening. The crack opening law should as far as possible be defined in terms of stress and displacement in order to remove some mesh sensitivity from the analysis. In some software's it is required that the crack opening law is defined in terms of stress and strain. In these cases, care should be taken since if the element size varies in the model, then also the fracture energy will vary for all elements (unless the fracture energy is defined separately for each element). In order to transfer the crack opening displacement to strain it should be divided with a representative length, a crack band width. It thereby corresponds to an assumption that the crack will localize on that length.

$$\varepsilon_{cr} = \frac{w_{cr}}{h}$$

where

ε_{cr} is the cracking strain

w_{cr} is the crack opening displacement [m]

h is the crack band length [m]

There are different opinions regarding the choice of the crack band length and it may affect the results significantly. Most are in agreement regarding the following cases

a) unreinforced concrete: $h = l_{element}$

b) Smeared reinforcement (in models where the element size is larger than the reinforcement spacing): $h = s_{rm}$

c) Separate reinforcement bars with a bond-slip definition: $h = l_{element}$

The choice of crack band width is also influenced by the type of element used (first or second order) and the shape of the element. For elements with irregular shapes, it is difficult to define one representative crack band width. Extra care should also be given to elements adjacent to symmetry lines, where $h = 2 \cdot l_{element}$ should be chosen for cracks parallel to the symmetry line.

Several different crack opening laws are available to define the descending branch of the nonlinear crack opening curve. The three most common curves are the following; linear, bilinear and exponential, and are illustrated in Figure 5-1.

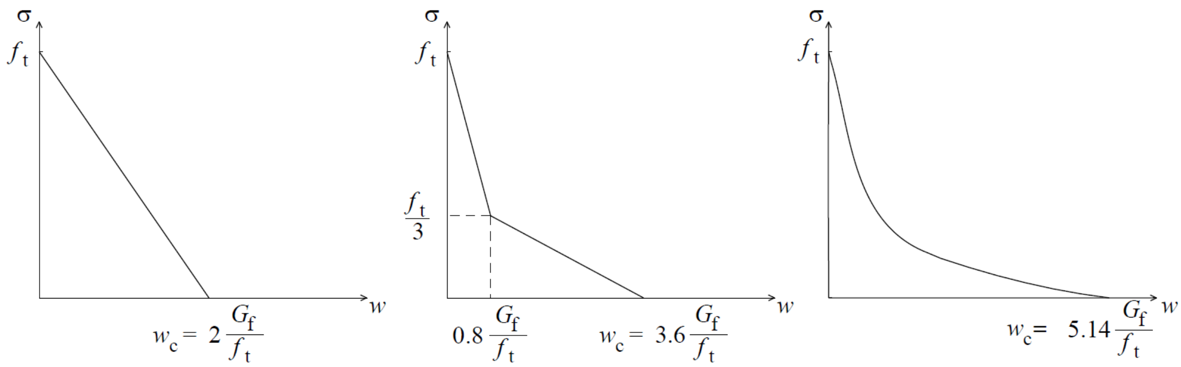


Figure 5-1: Linear, bilinear and exponential crack opening curves used in numerical analyses, from Malm (2009).

The equations to calculate the linear and bilinear curves are shown in the figure. The exponential curve was defined by Cornelissen et al. (1986) as

$$\frac{\sigma}{f_t} = f(w) - \frac{w}{w_c} f(w_c)$$

where $f(w)$ is a displacement function given by

$$f(w) = \left[1 + \left(\frac{c_1 w}{w_c}\right)^3\right] \exp\left(-\frac{c_2 w}{w_c}\right)$$

where

w is the crack opening displacement [m]

w_c is the crack opening displacement at which the crack can be considered as stress-free [m]

c_1 is a material constant and it is equal to 3.0 for normal density concrete

c_2 is a material constant and it is equal to 6.93 for normal density concrete

The linear is the simplest curve and the exponential is closest to the real behaviour of concrete. Their usage depends on the accuracy needed for the analysis. On a small scale, in one specific element, regardless of which curve is used all elements will crack at the same time. The difference is how fast the crack propagation will go, i.e. how fast the cracked element has to unload. The bilinear and exponential curve requires faster unloading directly after crack initiation (and thereby also smaller element size as seen in Section 4.2.2).

In Figure 5-2, the influence of fracture energy (G_f) and the tensile strength (f_{ct}) on the results is shown. As previously mentioned, cracking is initiated when principal maximum stress is equal to the tensile strength. The crack initiation can be observed the load-deformation curve, see graph (b), as the point when it is no longer linear. It can also be seen that the load capacity is higher than the cracking load.

As seen in the figure, increasing the tensile strength of the material will result in an increased load capacity of the structure, but at the same time if the fracture energy is constant, then a more brittle behaviour is obtained in the unloading response. In the figure, it can be seen that the unloading curve is steeper for the cases with higher tensile strength.

Increasing the fracture energy, also increase the structural load capacity but without reducing the brittleness of the structure, as seen in graph (c). In the figure, it can be seen that the unloading curves are parallel to each other, i.e. a more ductile behaviour is obtained.

In the last graph (d), the calculated response of the beam is shown for different crack opening laws. It can be seen in the figure that the linear curve, overestimates the strength and stiffness of unreinforced concrete beam, while the exponential function corresponds quite well to the experimental curves.

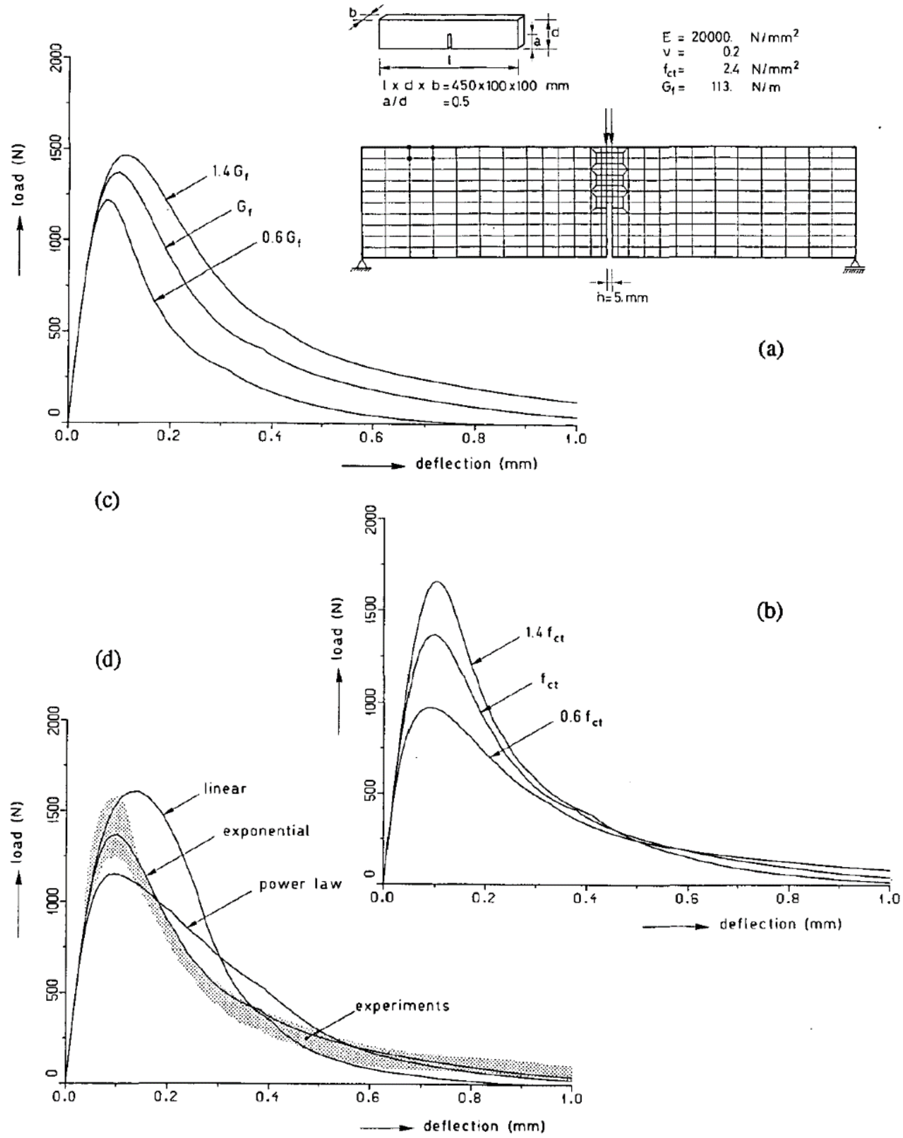


Figure 5-2: Influence of the crack opening laws and material properties on the response, from Rots (1988).

Uniaxial compressive behaviour

The 28 day compressive cylinder strength can be obtained from Table 3.1 in Eurocode 2 and the mean value varies between 20 - 98 MPa for concrete grades C12/15 to C90/105. In analyses of concrete structures, it is the cylinder strength rather than cube strength that should be used. In Sweden, it was only the cube strength that was given in older codes.

A uniaxial compressive curve is given in Eurocode 2 and shown in Figure 5-3 below. It is recommended to be used, unless measurements are available.

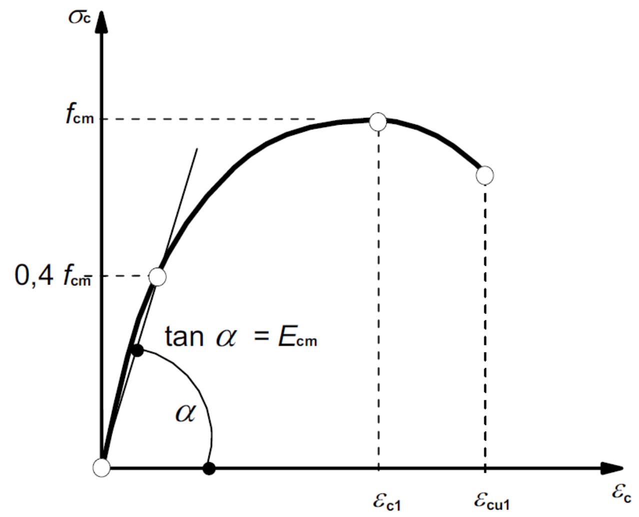


Figure 5-3 Uniaxial compressive curve according to Eurocode 2. From Eurocode 2 (2008).

The equation is as follows

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

where

$\eta = \varepsilon_c / \varepsilon_{c1}$

ε_{c1} is the strain at the maximum stress, $\varepsilon_{c1} = 0.8 f_{cm}{}^{0.31}$ in [‰]

$k = 1.05 E_{cm} \cdot \frac{|\varepsilon_{c1}|}{f_{cm}}$

The equation is valid for $0 \leq |\varepsilon_{c1}| \leq |\varepsilon_{cu1}|$, where ε_{cu1} is given in Table 3.1 in Eurocode 2 and is 3.5 ‰ for ≤C50/60.

In most (or all) FE-software's the program will assume that constant stress can be maintained for strains larger than the last point on the curve. Therefore, it is important to define that crushing of concrete occurs at a strain equal to ε_{cu1}. This is done by adding one additional point to the stress-strain curve where the stress is defined equal to zero (or close to zero[1]) for strains higher than ε_{cu1}. An example of a compressive curve based on the Eurocode equation, but discretized into nine points is shown in Figure 5-4.

[1] A value equal to 1 % of the compressive strength is considered sufficent.

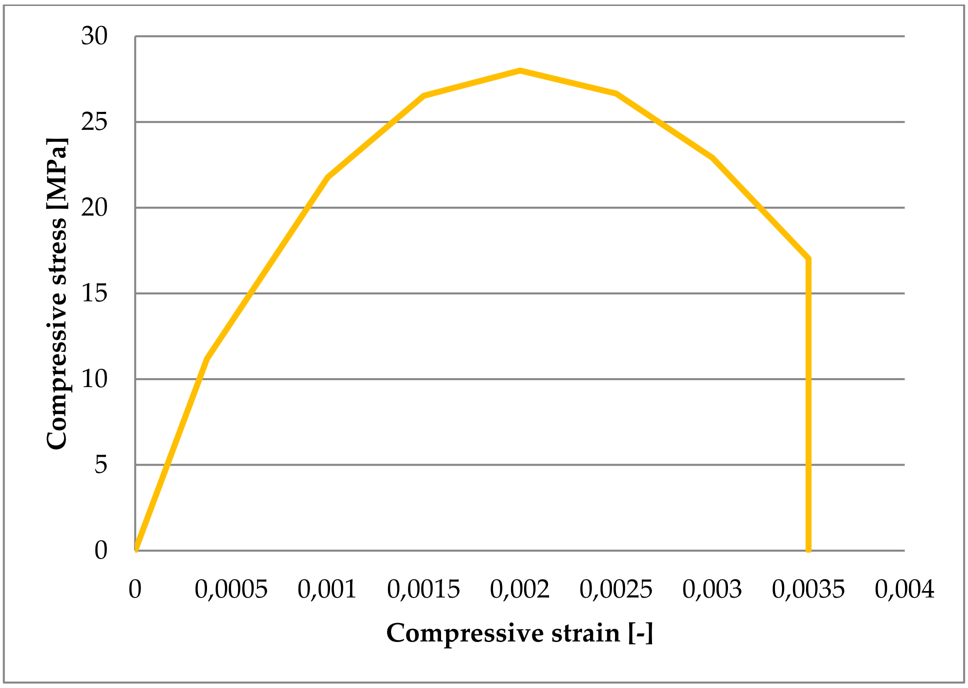


Figure 5-4 Example of a compressive curve for concrete.

Shear behaviour of a crack

In conventional normal strength concrete, cracks will be localized in the cement paste between the aggregates, as illustrated in Figure 5-5. The cracks thereby have an irregular surface and as a result of this; resistance for shear deformations.

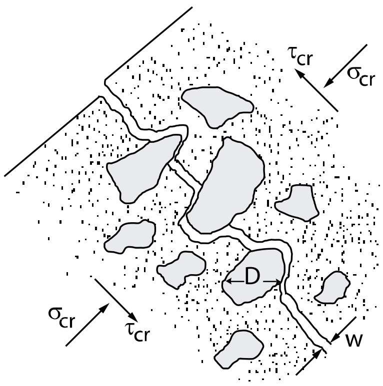


Figure 5-5 Illustration of a crack in concrete, from Malm (2006).

In traditional design based to design codes, the shear resistance of cracked structures is normally considered by the reduction of the shear modulus. A similar approach is often used in numerical analyses, for instance in the fixed crack approach as described in Section 5.3.1.

One expression to describe the degradation of the shear resistance due to cracking was derived by Kolmar (1986) and is for instance used in Cervenka et al. (2016). A shear retention factor ($0 \leq \beta \leq 1$) which depends on the cracking strain is used to reduce the shear modulus of concrete

$$\beta = c_3 \frac{-\ln\left(\frac{1000\varepsilon_u}{c_1}\right)}{c_2}$$

with

$$c_1 = 7 + 333(\rho - 0.005)$$

and

$$c_2 = 10 - 167(\rho - 0.005)$$

where

ρ is the reinforcement ratio [-], note that the equations above are defined valid for $0 \leq \rho \leq 0.02$. As default in Cervenka et al. (2016), the effect from reinforcement is not considered when calculating the shear retention factor.

c_3 is a user constant in Cervenka et al. (2016) and is by default assumed equal to one

The reduced shear modulus of the concrete material can thereby be defined as

$$G_{red} = \beta G$$

where,

G is the shear modulus for uncracked concrete, i.e. $G = \frac{E}{2(1+\nu)}$

In numerical models based on plasticity theory (see Section 5.3.1), a dilation factor is used to describe the shear resistance of cracked concrete. The dilation factor defines the amount of plastic volumetric strain developed during plastic shearing. The dilatation effect is illustrated for an element in Figure 5-6. In the left figure, a case where the dilatation angle is zero is illustrated, i.e. a shear force leads only to a shearing deformation, while the right figure shows a case where the dilatation angle is > 0.

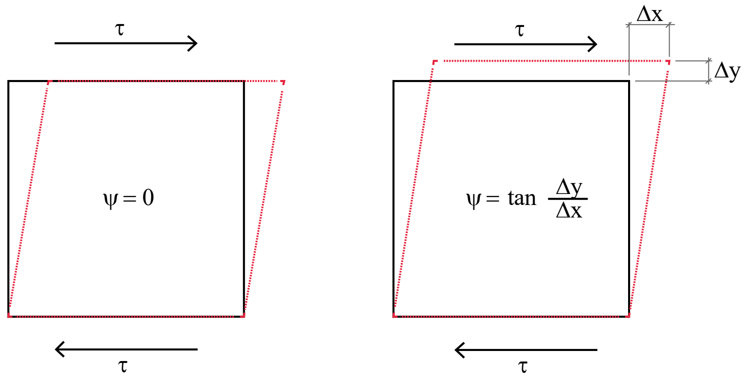


Figure 5-6 Influence of dilatation factor on an element, left – no dilatation, right – with dilatation.

Suggestions regarding the size of the dilatation angle at high confining pressures can be found in Malm (2009).

Multi-axial stress states

Concrete in uniaxial compression (and tension) is typically brittle in its nature; however, if concrete is subjected to high multiaxial compressive stresses (i.e. in 2D or especially in 3D) then its response becomes more ductile. A ductile material has a larger possibility of redistributing and counter-balance stress concentrations than a brittle material. The ductility of a material is related to the amount of energy that is consumed per deformation unit. In brittle materials small amount of energy is consumed per deformation unit, while larger amount of energy is consumed in ductile materials.

For instance, when concrete is subjected a biaxial stress-state (i.e. 2D) its strength (in compression and tension) changes according to Figure 5-7. In the case with biaxial compressive stresses, the compressive strength increase due to the confinement effect. For instance, when the compressive stresses in both direction are identical ($\sigma_1 = \sigma_2$), an increase in strength of typically 16 % is obtained (compared to the uniaxial compressive strength). For stress-states, where the concrete is subjected to tensile stress in one direction and compressive stress in the other direction, it can be seen in the figure that both the tensile and compressive strengths are reduced. This type of stress state occurs in cases with high shear stresses.

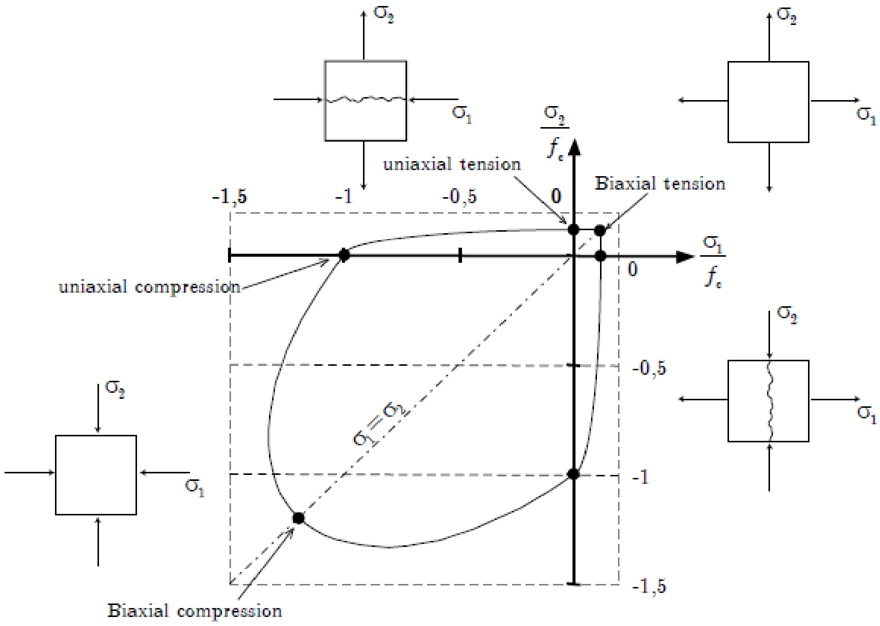


Figure 5-7 Failure envelope at biaxial stress states. From Malm (2006).

At triaxial stress-states, the compressive strength can increase significantly and the failure mode becomes more ductile as illustrated in Figure 5-8. In the figure, the compressive stress in the normal direction of the specimen is shown as a function of the displacement. Three curves are illustrated, one without confinement, i.e. $\sigma_2 = 0$, and the other two curves for increasing amount of confining pressure $\sigma_2/\sigma_1 = 0.2$ and $\sigma_2/\sigma_1 = 1.0$ respectively. For the uniaxial case, i.e. without confinement ($\sigma_2/\sigma_1 = 0$), the specimen will expand in the radial direction due to the poisson's ratio. If a confining pressure is present, this radial expansion is restrained and hence the compressive strength is increased significantly. According to Eurocode 2, the increase in compressive strength can be about 375 % if the concrete is subjected to equal compressive stresses in three directions ($\sigma_1 = \sigma_2 = \sigma_3$).

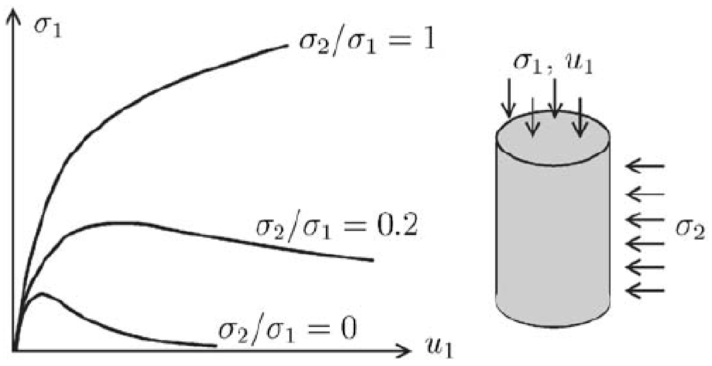


Figure 5-8 Influence of confinement, i.e. 3D stress states in compression. From Mang et al. (2003).

5.1.3 Hardening of concrete

Depending on the purpose of the analysis, different level of details may be required to analyse hardening of concrete and especially if young concrete is to be considered. In this section, the focus is on the development of the strength according to Eurocode 2. More detailed analyses of fresh and young concrete is considered out of the scope for this report.

Several design standards, such as Eurocode 2 and fib Model Code (2010) defines the development in strength to be dependent on a parameter β_{cc} that is dependent on the age of the concrete. The following equations are valid for a temperature equal to +20 ºC according to Eurocode 2. When using these equations for a different temperature state, the time should be replaced with an equivalent age. The equivalent age is further described in Section 5.1.5.

$$\beta_{cc} = \exp\left(s\left(1-\left(\frac{28}{t}\right)^{0.5}\right)\right)$$

where,

t is the concrete age (in days)

s is a coefficient that depends on the type of cement and is equal to

= 0.20 for cement in the strength classes CEM 42.5 R, CEM 52.5 N and CEM 52.5 R (class R)
= 0.25 for cement in the strength classes CEM 32.5 R, CEM 42.5 N (class N)
= 0.38 for cement in the strength classes CEM 32.5 N (class S)

Based on this time-function, the development of compressive and tensile strength over time can be determined as follows, according to Eurocode 2;

$$f_{cm}(t) = \beta_{cc}(t) \cdot f_{cm(28\,days)}$$

and

$$f_{ctm}(t) = \left(\beta_{cc}(t)\right)^{\alpha} \cdot f_{ctm(28\,days)}$$

where,

$\alpha = 1$ for $t < 28$

$\alpha = 2/3$ for $t \geq 28$

The growth of the tensile strength is not given in fib Model Code (2010). Instead it is mentioned in Model Code 2010 that it can be assumed that the relative increase in strength is similar for compressive and tensile strength in cases with moist curing conditions for time periods $t \leq 7$ days and for $t \geq 28$ days. For the interval in-between, residual stresses may lead to a temporarily reduction of the tensile strength (Model Code, 2010).

The elastic modulus, can according to Eurocode 2 be calculated as follows

$$E_{cm}(t) = \left(\frac{f_{cm}(t)}{f_{cm(28\ days)}}\right)^{0.3} \cdot E_{cm(28\ days)}$$

where,

$E_{cm}(t)$ is the time-dependent elastic modulus [Pa]

$f_{cm}(t)$ is the time-dependent compressive strength [Pa]

$f_{cm(28\ days)}$ defines the compressive strength at an age of 28 days [Pa]

$E_{cm(28\ days)}$ defines the elastic modulus at an age of 28 days [Pa]

When comparing the definition of the increase in elastic modulus in Model Code 2010 and Eurocode 2, the only difference is that the exponential is 0.3 in Eurocode 2 is instead 0.5 according to Model Code 2010. Thereby, the definition given in Model Code 2010 results a slower growth in elastic modulus compared to Eurocode 2.

The relative increase in strength and elastic modulus based on Eurocode 2 is shown in Figure 5-9. As it can be seen in the figure, the elastic modulus is increasing faster than both compressive and tensile strength. According to Eurocode 2, the relative increase in tensile and compressive strength is equal for time periods less than 28 days. After 28 days, it is assumed that the relative increase in tensile strength is slower than for the compressive strength.

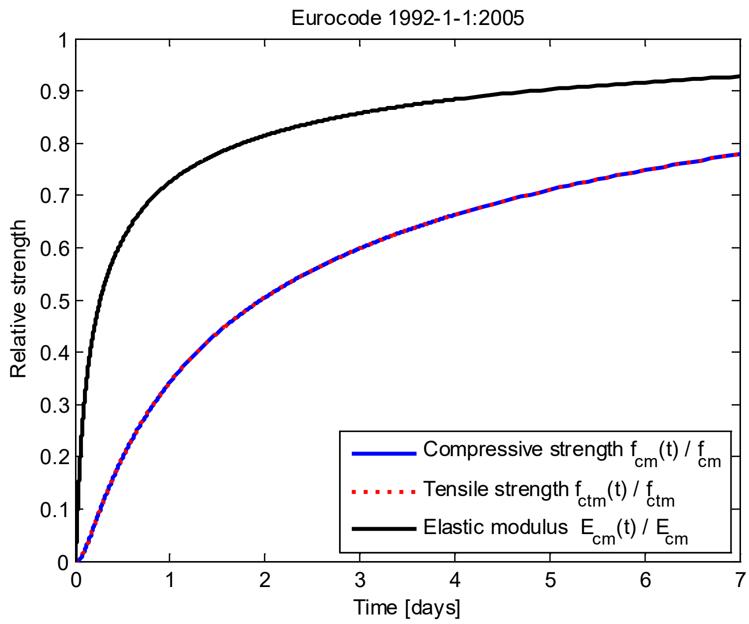


Figure 5-9 An example of increase in strength and elastic modulus based on Eurocode 2. From Blomdahl et al. (2016)

5.1.4 Shrinkage

Concrete shrinkage is an important factor when focusing on maintaining durable structures. Over time, the shrinkage may induce cracking which can significantly decrease the life expectancy of the concrete structure. (Holt, 2001).

Concrete shrinkage may be divided into early age and long-term behaviour, where the early age primarily refers to the first day after placement. The shrinkage may be a result from moisture loss (drying shrinkage) or internal reactions (autogenous shrinkage).

The early age shrinkage (and creep) is not covered in this report. The early age drying shrinkage may to a large extent be reduced by proper handling of the concrete for the first few hours after placement. (Holt, 2001).

Concrete hydraulic structures have in general massive cross-sections and in addition, parts of the structures may be subjected to water pressure. Thereby, the drying shrinkage is expected to be low compared to other types of civil structures such as houses or bridges. In design codes, the equations are developed based on an assumption of a constant shrinkage over the cross-section. This is a fairly poor estimate for most hydraulic structures. In hydraulic concrete structures, it is most likely that uneven (non-uniform) shrinkage occurs both over the sectional thickness but also between different parts with different thicknesses. Thereby, in order to perform a detailed numerical analysis of shrinkage it is required to perform an analysis of the drying of concrete, which can be converted into shrinkage strains in the structure.

In numerical analyses, shrinkage is in general treated as a load where a field of moisture content (or equivalent temperatures) are used as input in the mechanical analyses and converted into structural strain with the use of expansion coefficients. Therefore, in the following section, the material behaviour due to shrinkage is described and in Section 7.3.2, it is described how these may be included as loads in the FE-analysis.

Shrinkage according to Eurocode 2

This section focus on the autogenous and drying shrinkage as described in Eurocode 2 for long-term behavior. In Eurocode 2, the total shrinkage is defined as the summation of autogenous and drying shrinkage

$$\varepsilon_{cs} = \varepsilon_{cd} + \varepsilon_{ca}$$

where,

ε_{cs} is the total shrinkage

ε_{cd} is the drying shrinkage

ε_{ca} is the autogenous shrinkage

An example of the time dependent development of shrinkage is illustrated in Figure 5-10. As it can be seen in the figure, the autogenous shrinkage is only occurring for the first year/years, while the drying shrinkage can be on-going for many years.

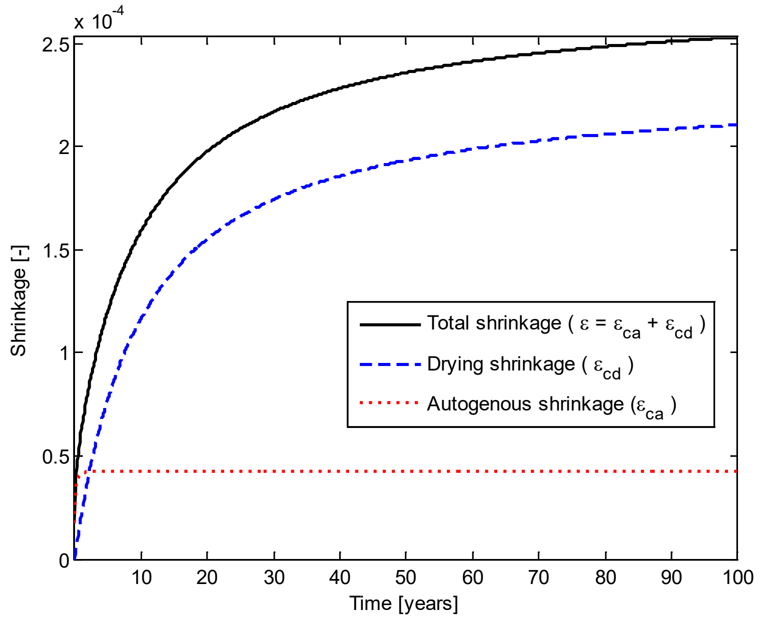


Figure 5-10 Example of time dependent development of shrinkage (total, drying and autogenous) based to Eurocode 2.

The time-dependent drying shrinkage may be expressed as

$$\varepsilon_{cd} = \beta_{ds}(t, t_0) k_h \varepsilon_{cd,0}$$

with

$$\beta_{ds}(t, t_0) = \frac{t - t}{(t - t_s) + 0.04\sqrt{h^3}}$$

where,

k_h is a coefficient that depends on the fictive sectional thickness h_0 and is equal to [1.0; 0.85; 0.75; 0.7] for $h_0 =$ [100; 200; 300; ≥500] mm.

t_s is the concrete age [days] when the shrinkage (or swelling) starts. This is normally the age of the concrete when the formwork is removed and/or when the after-treatment stops.

h_0 is the fictive sectional thickness of the cross-section [mm], $h_0 = {}^{2A_c}/_{u}$

A_c is the area of the cross-section [mm²]

u is the perimeter of the part of the cross-section that is subjected to drying [mm]

The final long-term shrinkage can according to Eurocode 2 be calculated based on the following expression

$$\varepsilon_{c,0d} = 0.85\left[(220 + 110 \cdot \alpha_{ds1})\exp\left(-\alpha_{ds2} \cdot \frac{f_{cm}}{f_{cm0}}\right)\right] \cdot 10^{-6} \cdot \beta_{RH}$$

with

$$\beta_{RH} = 1.55\left[1 - \left(\frac{RH}{RH_0}\right)^3\right]$$

where

f_{cm} is the average compressive strength [MPa]

f_{cm0} = 10 MPa

α_{ds1} is a coefficient that depends on the type of cement

= 3 for cement class S

= 4 for cement class N

= 6 for cement class R

α_{ds2} is a coefficient that depends on the type of cement

= 0.13 for cement class S

= 0.12 for cement class N

= 0.11 for cement class R

RH is the ambient relative humidity [%]

RH_0 = 100 %

The autogenous shrinkage can be determined as

$$\varepsilon_{ca} = \beta_{as}(t)\varepsilon_{ca}(\infty)$$

with

$$\varepsilon_{ca}(\infty) = 2.5(f_{ck} - 10) \cdot 10^{-6}$$

and

$$\beta_{as}(t) = 1 - \exp\left(-0.2\sqrt{t}\right)$$

In this approach, the final drying shrinkage is based on a fictive sectional thickness for the concrete member, i.e. the whole structure is assumed to reach the same final drying shrinkage. In addition, the equations are based on a relative humidity for the ambient air that the concrete is assumed to reach after infinite time. As mentioned previously, if the structural member is subjected to one-sided water pressure, then the drying shrinkage will be rather limited.

Thereby, as mentioned previously, for hydraulic structures it may be required to perform more detailed analyses than based on these equations. For instance, Malm et al. (2013b) showed that a concrete structure in a power house which was acting as support for the generator still had a relative humidity of 95 % in the central parts of the concrete member after 20 years. It should be noted that this structure had no direct contact with water, so the reason for the high moisture content in the concrete was the slow drying process and the large thickness of the cross-section.

Calculating drying shrinkage based on moisture content

When calculating drying of concrete, the moisture content or the relative humidity are often defined as the driving potential. It is also possible to use other alternatives such as pore pressure, water saturation etc. The moisture transport is influenced by the temperature conditions, however, this is in most cases neglected when analysing the drying shrinkage.

In the following section, an example on how drying shrinkage can be simulated with FE-analyses based on input from moisture transport analyses is given. There are many references on this topic, but the level of detail in these approaches are often too high and thereby these methods are difficult to use on large structures.

One simple approach to calculate the moisture transport in concrete due to diffusion of water vapour is given in Section 5.1.12.2.1 in fib Model Code (2010).

In this section, one approach to calculate the shrinkage based on moisture content is presented. Approaches based on relative humidity as the driving potential can for instance be found in Gasch et al. (2016) or Malm et al. (2013b).

Assuming that a moisture transport analysis has been conducted where the results includes moisture content in each node or element. These moisture contents can then be imported into the mechanical analysis, by converting the moisture content to corresponding drying shrinkage strain, for instance as defined by Lundquist and Nilsson (2011)

$$\varepsilon_{\mathrm{sh}}(x,y,z,t) = \frac{w_{\mathrm{i}} - w(x,y,z,t)}{w_{\mathrm{i}} - w_{\infty}} * \varepsilon_{\mathrm{sh}.\infty}$$

where,

$\varepsilon(x,y,z,t)_{\mathrm{sh}}$ is the shrinkage strain at time t in the concrete node located at (x,y,z)

w_{i} is the initial moisture content [kg/m^3]

w_{∞} is the equilibrium moisture content [kg/m^3]

$w(x,y,z,t)$ is the moisture content at time t in the concrete node located at (x,y,z).

$\varepsilon_{\mathrm{sh},\infty}$ is the final shrinkage strain

5.1.5 Creep

Creep is defined as increased (inelastic) strains for sustained loads. The strain increases with highest rate early in the loading period and the rate decreases with time. Creep is a highly complicated phenomenon and it is influenced by moisture and temperature variations, the size of the structural member, age of the concrete at loading, the concrete composition and the size and duration of the load.

Creep according to Eurocode 2

According to Eurocode 2, if the compressive stresses due to the sustained loads are less than 45 % of the compressive strength at the time when the sustained load is applied, then it is sufficient to consider only linear creep.

The creep deformation $\varepsilon_{\mathrm{cc}}(\infty, t_0)$ at the time $t = \infty$ for a case with constant compressive stress σ_{c} (applied when the concrete had an age t_0), may be determined as follows according to Eurocode 2

$$\varepsilon_{\mathrm{cc}}(\infty, t_0) = \varphi(\infty, t_0) \cdot \left(\frac{\sigma_c}{E_c}\right)$$

where,

E_c is the tangent elastic modulus $E_c = 1.05 E_{cm}$

An example of time dependent creep factors for different ages at loading (based on Eurocode 2, i.e. the equations in shown later in this section) is shown in Figure 5-11.

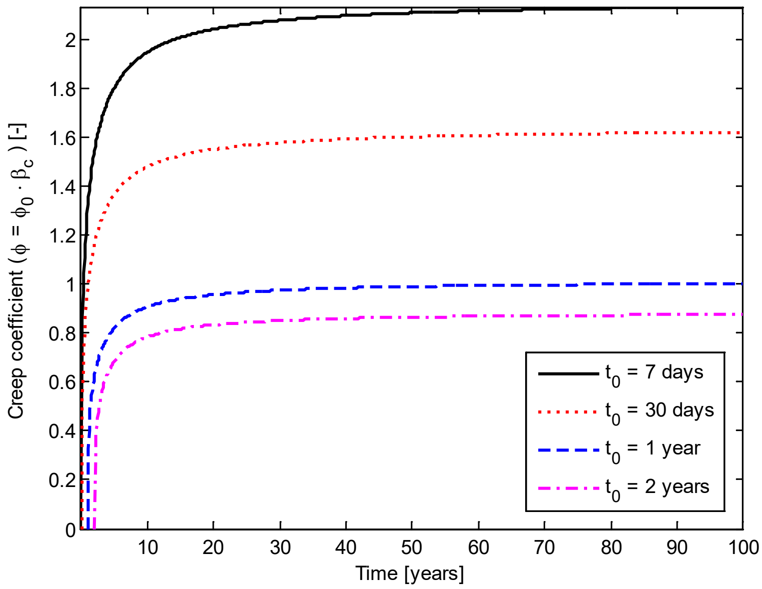


Figure 5-11 Example of time dependent creep factors for different ages at loading based to Eurocode 2.

The creep factor $\varphi(t, t_0)$ can be calculated as follows, according to Appendix B in Eurocode 2

$$\varphi(t, t_0) = \varphi_0 \cdot \beta_c(t, t_0)$$

where,

φ_0 is the nominal creep factor, and can be estimated as

$$\varphi_0 = \varphi_{RH} \cdot \beta(f_{cm}) \cdot \beta(t_0)$$

where,

φ_{RH} is a factor that considers the influence of relative humidity and is equal to

$$\varphi_{RH} = 1 + \frac{1 - \mathrm{RH}/100}{0.1\sqrt[3]{h_0}} \text{ for } f_{cm} \leq 35 \text{ MPa}$$

or

$$\varphi_{RH} = \left[1 + \frac{1 - \mathrm{RH}/100}{0.1\sqrt[3]{h_0}} \cdot \alpha_1\right] \cdot \alpha_2 \text{ for } f_{cm} > 35 \text{ MPa}$$

where,

RH is the relative humidity [%]

$\beta(f_{cm})$ is a factor that accounts for the compressive strength, $\beta(f_{cm}) = {}^{16.8}\!/\!\sqrt{f_{cm}}$

f_{cm} is the mean compressive strength after 28 days [MPa]

$\beta(t_0)$ is a factor that accounts for the concrete age at loading (t_0),
$\beta(t_0) = {}^{1}\!/\!(0.1 + t_0^{0.20})$

h_0 is the fictive sectional thickness of the cross-section [mm], $h_0 = {}^{2A_c}\!/\!u$

A_c is the area of the cross-section [mm²]

u is the perimeter of the part of the cross-section that is subjected to drying [mm]

$\beta_{cc}(t, t_0)$ is a coefficient that describes the time-dependent development after loading and is calculated as follows

$$\beta_{cc}(t, t_0) = \left[\frac{(t - t_0)}{\beta_H + t - t_0}\right]^{0.3}$$

t is the concrete age [in days]

t_0 is the concrete age at the time of loading [in days]

β_H is a factor that accounts for the relative humidity and the structural members fictive sectional thickness as follows

$$\beta_H = 1.5[1 + (0.012\mathrm{RH})^{18}]h_0 + 250 \ \leq 1500 \text{ for } f_{cm} \leq 35 \text{ MPa}$$

or

$$\beta_H = 1.5[1 + (0.012\mathrm{RH})^{18}]h_0 + 250 \cdot \alpha_3 \ \leq 1500 \text{ for } f_{cm} \geq 35 \text{ MPa}$$

$\alpha_1, \alpha_2, \alpha_3$ are coefficients that accounts for the concrete strength as follows

$$\alpha_1 = \left[\frac{35}{f_{cm}}\right]^{0.7}$$

$$\alpha_2 = \left[\frac{35}{f_{cm}}\right]^{0.2}$$

$$\alpha_3 = \left[\frac{35}{f_{cm}}\right]^{0.5}$$

The influence from type of cement used is considered by modifying the age at loading, i.e. t_0, as follows

$$t_0 = t_{0,T} \cdot \left(\frac{9}{2 + {t_{0,T}}^{1.2}} + 1\right)^{\alpha} \geq 0.5$$

where,

$t_{0,T}$ is the temperature adjusted value of the concrete age at loading [days]

α is a coefficients that accounts for type of cement

=-1 for cement class S

= 0 for cement class N

= 1 for cement class R

The influence of increased or reduced temperature on the maturity degree of concrete can be considered by adjusting the concrete age as follows

$$t_T = \sum_{i=1}^{n} \Delta t_i e^{\left(13.65 - \frac{4000}{273+T(\Delta t_i)}\right)}$$

where,

t_T is the temperature adjusted concrete age that replaces t in all expressions [in days]

$T(\Delta t_i)$ is the temperature in the time interval Δt_i [in °C], the equation above is valid within the temperature range 0 to 80 °C

Δt_i is the number of days for which the temperature T occurs.

5.2 STEEL

In linear elastic analyses, the same material properties needed to define the linear behaviour of concrete (see Section 5.1.1) are required also for steel. Below, suitable material properties for reinforcement and tendons are given.

5.2.1 Reinforcement

Unless more detailed information are available, the density and elastic modulus of reinforcement may be assumed equal to 7850 kg/m^3 and 200 GPa respectively.

For cases when the nonlinear behaviour of reinforcement is included, a von Mises yield criteria is normally assumed for reinforcement. According to Eurocode 2, a bilinear stress-strain curve can be assumed unless actual test data are available. As a conservative approach, the strain hardening of reinforcement can be neglected. However, in this report the recommended curve includes strain hardening and also describes a possible rupture. An example of a stress-strain curve for a hot-rolled bar is shown in Figure 5-12 a) and the idealized curve in Eurocode 2 is illustrated in Figure 5-12 b).

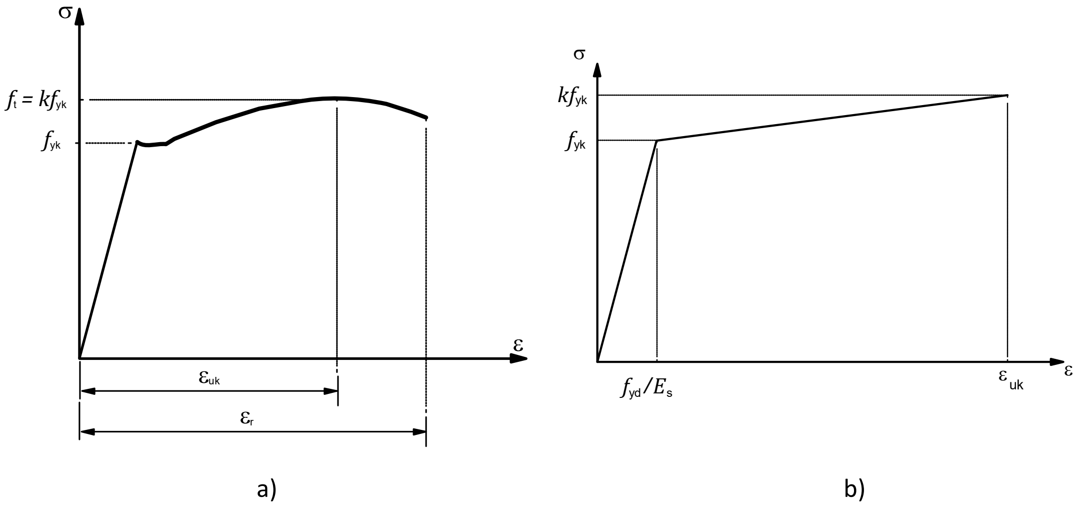


Figure 5-12 a) Stress-strain curve for reinforcement, b) idealized curve according to Eurocode 2.

Reinforcement of type B500B, is most the most frequently used type of reinforcement in Sweden. A typical stress-strain curve for this type of reinforcement is shown in Figure 5-13.

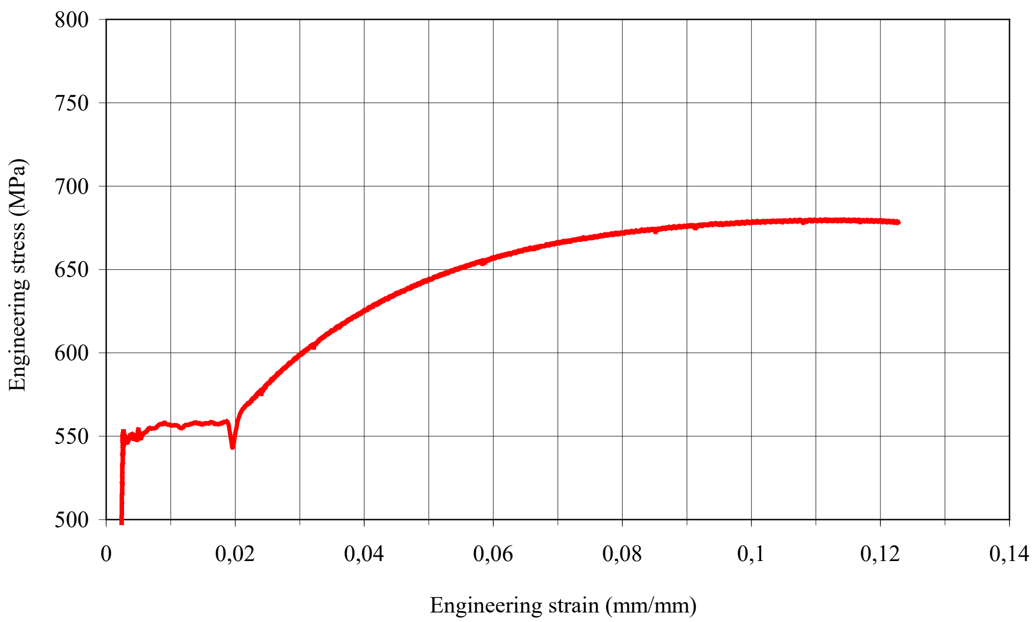


Figure 5-13 Typical stress-strain curve for reinforcement B500B, from Malm (2015)

Based on Eurocode 2, the following characteristic material properties can be assumed for the idealized curve shown in Figure 5-14.

- Elastic modulus 200 GPa
- Yield strength $f_{yk} = 500$ MPa

- Ultimate strength (at least) $f_t = k \cdot f_{yk} = 540$ MPa (k ≥ 1.08). (The absolute maximum permissible value is $f_t \leq 650$ MPa)
- Elongation at maximum load $\varepsilon_{uk} = 5$ %

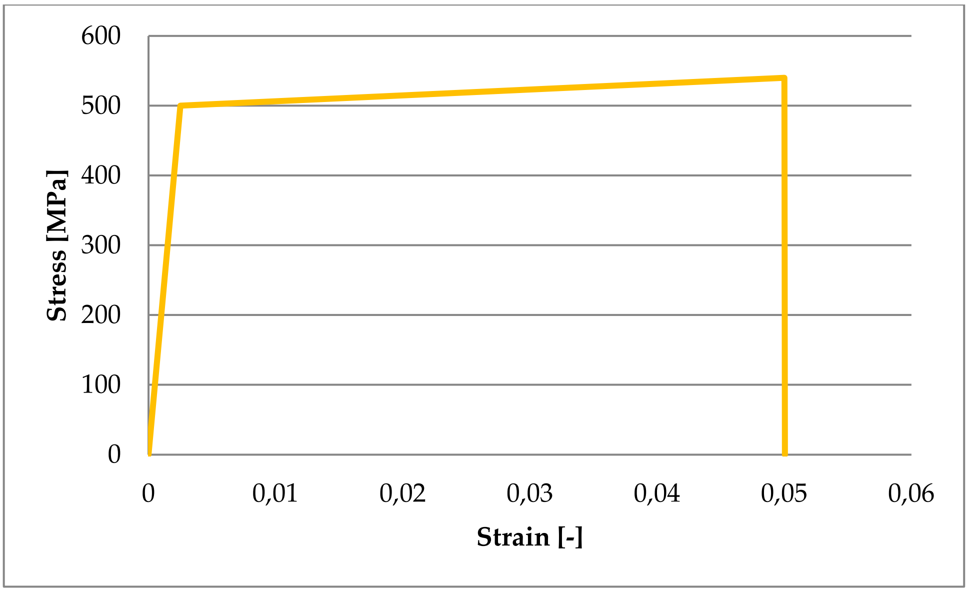


Figure 5-14 Idealized characteristic stress-strain curve for B500B based on Eurocode 2.

5.2.2 Tendons

Unless more detailed information are available, the density of tendons (strands, bars or wires) may be assumed equal to 7850 kg/m^3.

The elastic modulus of pre-stressed bars or wires varies in general between 195 to 205 GPa and unless more detailed information are available, 205 GPa can be assumed as the design value.

The elastic modulus of pre-stressed strands varies in general between 185 to 205 GPa and unless more detailed information are available, 195 GPa can be assumed as the design value.

A typical stress-strain curve for a tendon is shown in Figure 5-15 a) and the corresponding idealized curve from Eurocode 2 is illustrated in Figure 5-15 b).

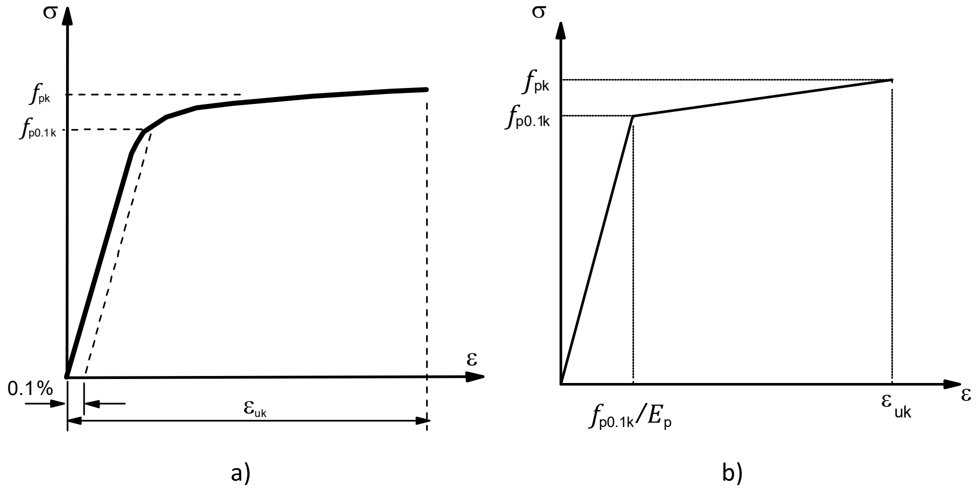


Figure 5-15 a) Stress-strain curve for typical tendon steel, b) idealized curve according to Eurocode 2.

An example of an idealized stress-strain curve for VSL 6-19 tendons is shown in Figure 5-16 with

- Elastic modulus 195 GPa
- Yield strength $f_{p0.1k}$ = 1640 MPa
- Ultimate strength f_{pk} = 1860 MPa
- Elongation at maximum load ε_{uk} = 2.5 %

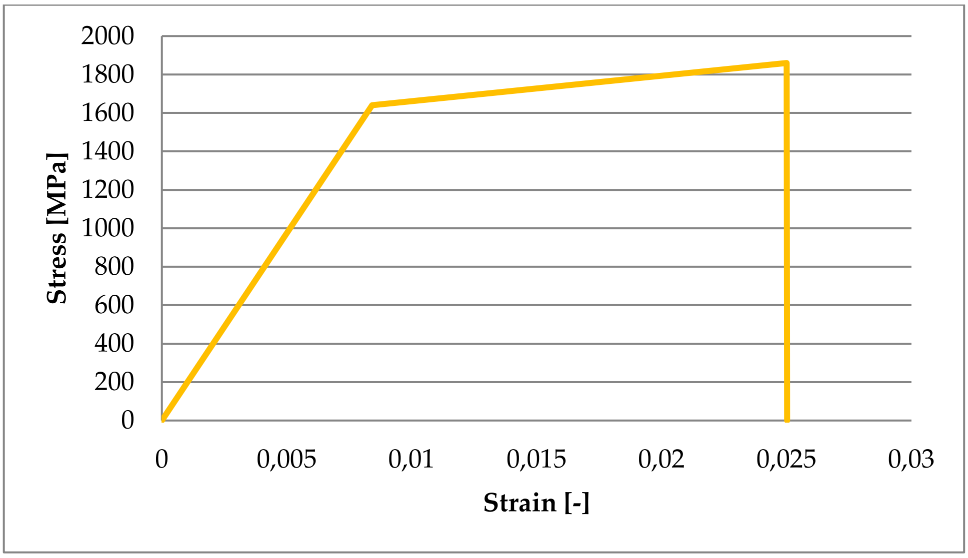


Figure 5-16 Example of a stress-strain curve for a tendon.

5.3 MATERIAL MODELS TO DESCRIBE THE NONLINEAR BEHAVIOUR OF CONCRETE

Material nonlinearity of concrete, due to cracking and crushing, have a significant influence on the structural behaviour. In this section, a brief description of common types of material models and their characteristics used to describe the nonlinear material behaviour of concrete is presented. It is out of the scope of this report to describe these theories in detail, and the interested reader may for instance read the following literature Karihalo (2003), Bangash (2001), Mang et al. (2003) among others.

There are several different theories on how to describe the nonlinear material behaviour of concrete in numerical analyses such as the finite element method. These are primarily defined based on one or several of the following theories; fracture mechanics, plasticity theory and/or damage theory. These theories has at least one thing in common and that is that cracking of concrete is described by a softening behaviour, i.e. a reduction of the stiffness.

In general, two main approaches can be used to describe crack propagation in concrete. Cracks can either be considered in a continuum approach where they distributed (smeared) over the elements or considered as discrete cracks which describes a physical separation of the two crack surfaces, as illustrated in Figure 5-17.

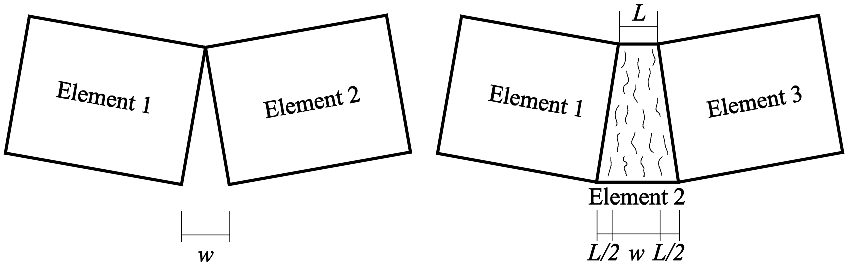


Figure 5-17: Discrete and smeared crack model. From Malm (2006).

Numerical simulations of crack initiation and propagation in concrete was initiated in the late 1960s by Ngo and Scordelis (1967) and Rashid (1968), who introduced discrete crack and smeared crack models, respectively. The discrete crack approach is focused to simulate initiation and propagation of dominate cracks. The smeared crack approach on the other hand is based on an assumption that many small cracks nucleate and only at a late stage of the loading process form one or more dominate cracks and therefore it describes the deterioration process through a constitutive relation by smearing the cracks over a continuum. (de Borst et al 2004)

The smeared crack approach was first introduced by Rashid (1968) and Cervenka and Gerstle (1971, 1972) and the discrete crack approach was first introduced to concrete structures by Saouma and Ingraffea (1981).

5.3.1 Smeared crack approaches

This means that in the smeared crack approach, no individual cracks are calculated but instead the strain in these elements is the sum of two parts; one part of the nonlinear behaviour of the crack/cracks and the second part is the elastic strain of the uncracked part of the concrete within the element. If the element size in one model based on smeared crack approach is larger than the characteristic crack spacing (which can be the case for reinforced structures), then the cracking strain in each elements represent two or more cracks. In the same way, if the element size is small enough then, normally, there will be uncracked elements between two cracked elements (depending on geometry and load) and thereby it is possible to also capture the crack spacing.

The smeared crack approach is by far the most common technique when used to analyse material nonlinearity of concrete in large structures and most available FE codes have at least one material model based on the smeared crack approach

One downside with smeared crack approaches are that they may be sensitive to the mesh and have a mesh dependency, this especially if the material behaviour is given in terms of stress and strain.

Fracture mechanics

In the smeared crack method, cracks appear in the integrations points of the element and their effect is distributed over the whole element. The total strain in one element (ε_{tot}) may therefore consist of the elastic part from the uncracked concrete (ε_{el}) and the nonlinear part from the crack opening (ε_{cr}), i.e.

$$\varepsilon_{tot} = \varepsilon_{el} + \varepsilon_{cr}$$

The crack opening strain is defined as the crack opening displacement divided by the crack band length (see Section 5.1.2), i.e.

$$\varepsilon_{cr} = \frac{w}{h}$$

In the smeared crack method, two different approaches exist to describe the propagation of crack; fixed crack model and the rotated crack model. The principal difference between the two models is illustrated in Figure 5-18.

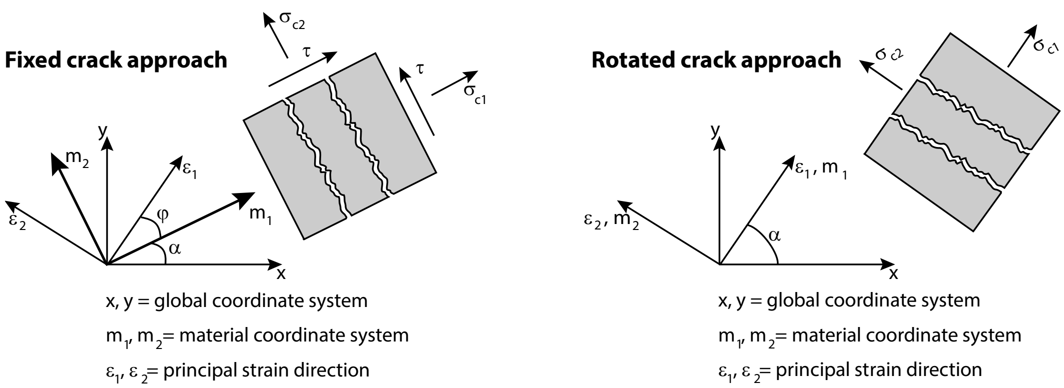


Figure 5-18 Fixed and rotated smeared crack approach respectively, reproduced from Cervenka et al. (2016).

In the fixed crack approach, the crack initiates in the element integration point when the maximum principal stress is equal to the tensile strength. The crack opening direction is equal to the maximum principal direction. During subsequent loading this crack direction is maintained, regardless of how the stress-states changes. Since the crack direction is different from the principal stress direction, shear stresses will be present at the crack surface. The shear resistance of cracked concrete, was discussed previously in Section 5.1.2. In the fixed crack approach, the shear capacity of a cracked element is described by the use of a shear retention factor β. The choice of shear retention factor may have a significant influence of the results.

If the stress changes orthogonal secondary cracks can occur in the same integration point, as a maximum three cracks per integration point (in a 3D problem).

In the rotated crack approach, the crack initiates in the element integration point when the maximum principal stress is equal to the tensile strength, just as in the previous approach. However, the difference is that if the subsequent stress state changes, the crack direction will rotate. The crack direction will always follow the principal stress/strain direction. Since the crack direction rotates to follow the principal stress direction, no shear stresses will occur at the crack plane. Only the following to stresses will occur in the crack plane; maximum compressive stress in the length direction of the crack (perpendicular to the crack width) and maximum principal tensile stress that causes crack opening (in the crack width direction)

The relationship between stress and strain in the material can be defined as follows

$$\boldsymbol{\sigma} = \mathbf{D}^s \boldsymbol{\varepsilon}_{ns}$$

where

$\boldsymbol{\sigma}$ is the stress vector

$\mathbf{D}^s$ is the secant stiffness

$\boldsymbol{\varepsilon}_{ns}$ is the strain vector

For a case with plain stress and where the poisons ratio is assumed equal to zero, the stress and strain may be defined as

$$\boldsymbol{\sigma} = (\sigma_{nn}, \sigma_{ss}, \sigma_{ns})^{\mathrm{T}}$$

where $\sigma_{ns} = \tau$ and

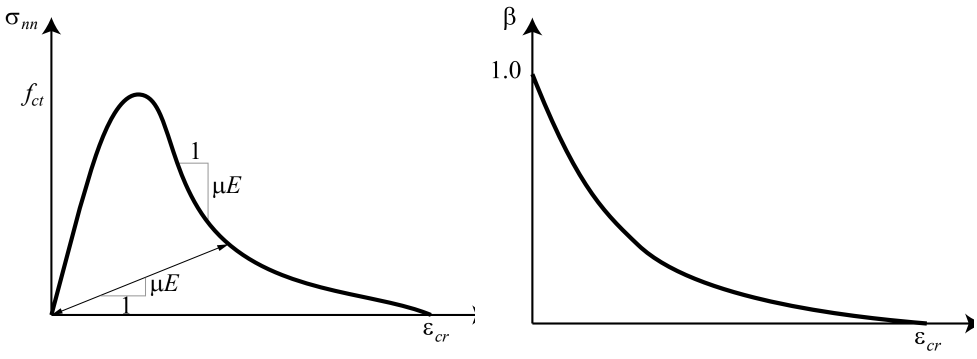
$$\boldsymbol{\varepsilon} = (\varepsilon_{nn}, \varepsilon_{ss}, \varepsilon_{ns})^{\mathrm{T}}$$

and where the secant modulus can be defined as follows for a model based on the fixed crack approach

$$\mathbf{D}^s = \begin{bmatrix} \mu E & 0 & 0 \\ 0 & E & 0 \\ 0 & 0 & \beta G \end{bmatrix}$$

The term μE describes the tension softening in the normal direction for successively increased cracking, starting from crack initiation with micro cracks and to the development of a macro crack, as illustrated in Figure 5-19 a). (de Borst et al. 2004)

The term βG describes the shear resistance, which is reduced for increasing cracking strain, as illustrated in Figure 5-19 b). A suitable choice of a shear retention factor was previously given in Section 5.1.2.



a) b)

Figure 5-19 a) Unaxial tensile stress – strain curve, b) Shear retention factor.

If the model instead, is based on a rotating crack approach, there is no shear strain along the cracks. However, in order to ensure co-axiality between the principal strain and the material axes, a tangent shear modulus G_t is defined, Cervenka et al. (2016). This could for instance be according to Crisfield and Willis (1989) as

$$G_t = \frac{\sigma_{nn} - \sigma_{ss}}{2(\varepsilon_1 - \varepsilon_2)}$$

One important limitation with this model is that no plastic strain/deformation is assumed to occur at unloading where a secant unloading approach is used. Thereby, the cracks may close completely at unloading.

When using both fixed and rotated crack approaches, there is a risk that stress-locking occurs which result in that the structural stiffness is overestimated after crack initiation. This phenomenon is described by Rots (1988) and was also shown in Malm (2009) where the structural deformation was underestimated significantly for these types of models. This phenomenon is in due to displacement compatibility between the uncracked (elastic elements) and the cracked elements, where the contribution of the elastic elements is overestimated since it does not properly represent the stress rotations.

In the rotated crack approach, there is a risk of spurious rotation of the crack direction that leads to convergence difficulties or in some cases, the crack may be able to find an alternative (false) rotated state of equilibrium and thereby allowing for higher loads and in this way overestimates the load capacity, Malm (2006).

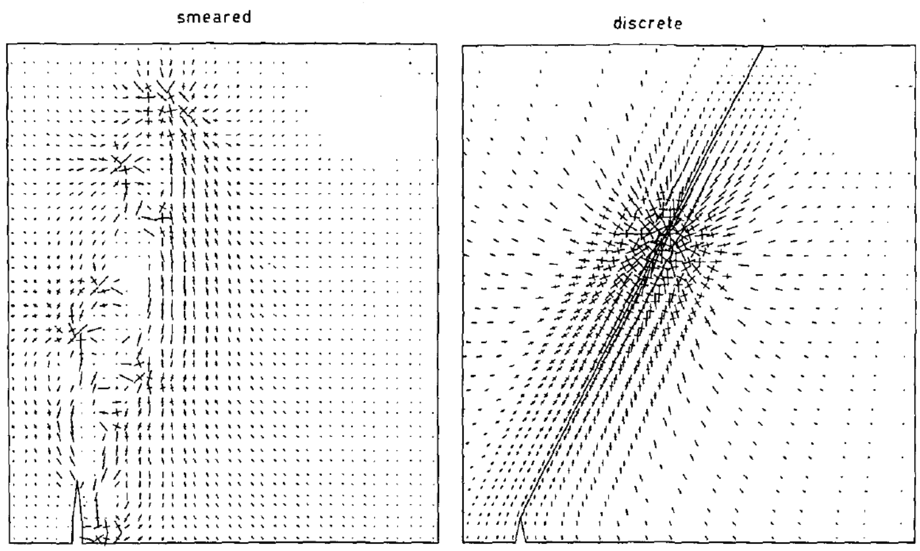


Figure 5-20 Principal stress trajectories for smeared (rotating) crack and discrete crack results, from Rots (1988).

One advantage with material models based on fracture mechanics is that they, in general, are based on few model parameters to define the material behaviour. Most of these model parameters correlates to actual physical material properties, which helps the definition of the material model.

Damage based models

In models based on damage mechanics, a damage parameter (d) is introduced which describes the damage evolution (i.e. cracking). A damage parameter equal to zero corresponds to an intact (uncracked) material while a damage parameter equal to one corresponds to a fully cracked material. This can in a simplified manner be illustrated in Figure 5-21 as the cross-section of the material is assumed to decrease due to cracking, where the damage parameter is calculated as the ratio of the remaining cross-section ($A_0 - A_i$) and the original uncracked area.

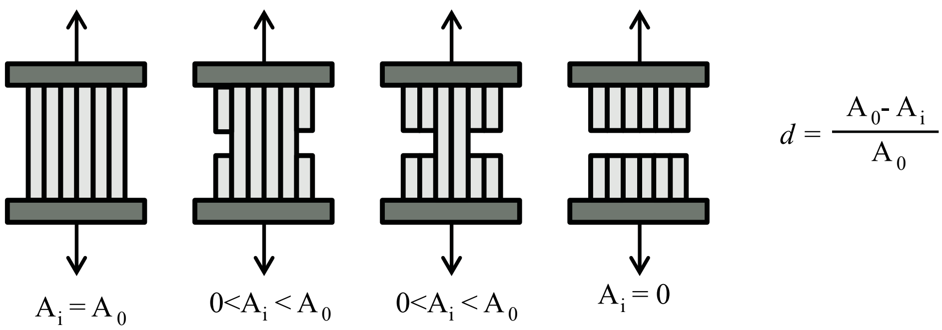


Figure 5-21 Damage model and evaluation of the damage parameter, reproduced from Jirasek (2014).

In the simplest form of damage theory, the damage parameter is considered as isotropic. This means that when a crack occurs, the stiffness is reduced with the factor $(1 - d)$ in all directions

$$\mathbf{D}^s = (1 - d)\mathbf{D}^0$$

where

$\mathbf{D}^s$ is the secant stiffness

d is the damage parameter

$\mathbf{D}^0$ is the initial stiffness

The initiation of the damage parameter is controlled by a failure criteria (or failure surface), which depends on the choice of damage model could for instance be the principal strain reaches the crack initiation strain.

An approach with isotropic damage without distinction between compressive and tensile is not suitable for concrete, and therefore it is common that two separate damage parameters are defined for tension (d_t) and compression (d_c) respectively. Another common, alternative approach for concrete material models is that damage theory is only applied to describe the tensile behaviour of concrete where for instance plasticity theory may be used to describe the compressive behaviour.

An isotropic damage model with damage evolution for tensile loads only is similar to a rotating crack approach described earlier, in the sense that a crack is weakening the element in all directions and therefore it does not matter if the principal direction changes since the material is still considered to be damaged in the principal direction.

One advantage with damage models is that they are simple and does not require iterations of internal material variables and therefore easily obtain a converging solution. There are also some more complex damage models, such as anisotropic damage models, where the damage is defined in orthogonal directions and thereby is the equivalent to a fixed crack approach.

Another advantage is that they, in general, are based on few model parameters to define the material behaviour. Most of these model parameters correlates to actual physical material properties, which helps the definition of the material model.

Plasticity based models

Plasticity theory is mainly used to describe ductile material behaviour but it is possible to use it to describe quasi-brittle materials.

In plasticity theory, the strain can be divided into two terms, one for the elastic strain (ε_{el}) for the intact part and one for the plastic strain (ε_{pl}) describing cracking or crushing. Models based on plasticity theory have many similarities with damage models, for instance a yield surface which may be similar to the failure surface in damage theory. The yield surface is used to describe the strength in multiaxial stress-states and is initially defined to correspond to the elastic limit. In the plasticity theory this yield surface will successively change in size as the material is subjected to nonlinear behaviour, for instance decrease in the tensile regime if cracking occurs, or expand in the compressive regime due to hardening. The parameter controlling the volumetric change of the yield surface is described with an internal scalar variable that permits a history dependence of the strain in the material, called the hardening parameter. In this case, the yield surface can expand or shrink (which is needed to describe the reduced strength and stiffness due to cracking) in stress-space, which is called as isotropic hardening. The coupling between the yield surface and the stress-strain relationship is determined through a flow rule. The flow rule gives an expression for the evolution of the increment of the plastic strain. (Malm 2009)

One important difference between plasticity models and damage models is that a permanent plastic strain occurs in the material after unloading in plasticity-based models, as illustrated in Figure 5-22. At unloading with a plasticity-based model, the stiffness at unloading is equal to the elastic modulus of the intact material (i.e. it has same inclination as the initial loading up to the tensile strength).

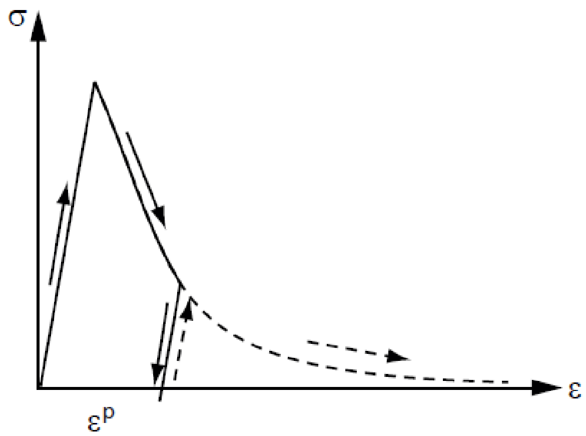


Figure 5-22 Elastic unloading with permanent plastic strain.

Even though plasticity based models may be defined as relatively simple, more detailed and advanced definitions are required to be able to implement these for concrete structures. When solving the response of a plasticity based material model, iterations are required of internal variables (i.e. hardening) which is not required in for instance damage models. Thereby, plasticity based models are in general more complicated to used to obtain a converging solution. In addition, some model parameters may be needed as input which are not normally measured in conventional concrete and therefore may be more difficult to determine.

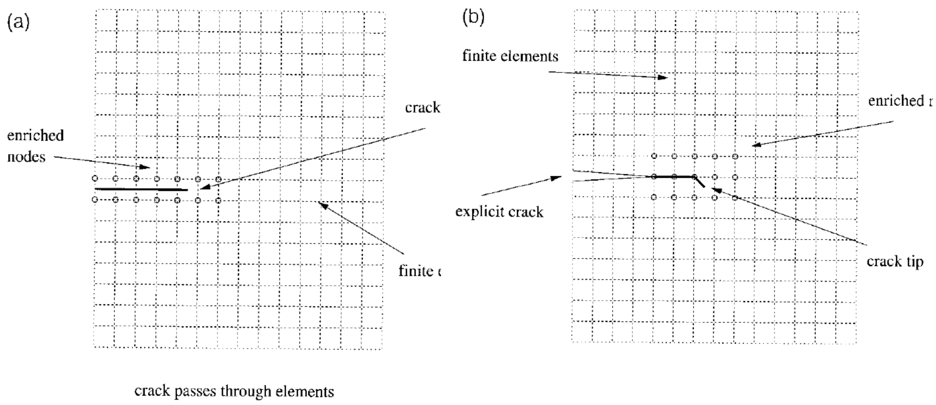
5.3.2 Discrete crack approaches

In the discrete crack approach, cracks are typically initiated in the intersection of two adjacent elements and create a physical distance (crack opening) between these elements. This can for instance be obtained with special purpose, interface, elements or constraints/interactions between these elements. The discrete crack approach is intuitively appealing since the crack is introduced as a geometric entity and its behaviour is defined in terms of stress and displacements (crack width).

In the first use of the discrete crack approach by Ngo and Ingraffea (1967) it was implemented in a numerical model where the propagation of a crack was defined to occur if the node ahead of the crack tip was subjected to a nodal force which corresponded to the tensile strength of the material. After this, the node was split into two nodes and the crack could continue to propagate further. (de Borst et al. 2004)

Thereby, one significant difference compared to the smeared crack approach (except the way a crack is represented) is that the bulk material (i.e. concrete) can be described with linear properties only while the nonlinear behaviour of the crack is defined only for the interface elements.

In the discrete crack approach, since cracking only occurs in the interface between ordinary concrete elements, this method is thereby subjected to mesh bias. This effect can, however, be reduced (if not eliminated) with the use of an adaptive meshing technique in order to iteratively remesh the model based the calculated results. In addition, there are more recent methods such as extended FEM (XFEM) developed by Belytschko and Black (1999) where discrete cracks can initiate within the elements and split them into two new elements, with the use of enriched nodes (or phantom nodes) that are activated as the tensile strength is exceeded in the model, as illustrated in Figure 5-23. This technique was for instance applied to an arch dam, in the paper by Goldgruber and Malm (2014) and by Goldgruber (2015).



a) b)

Figure 5-23 Two enriching strategies near the crack, a) the elements and nodes surrounding the whole crack as it passes through are enriched, b) the mesh conforms to part of the crack and enrichment is only near the crack tip including the last mesh conforming node, from Belytschko and Black (1999)

Fracture mechanics

The nonlinear crack opening behaviour of concrete is commonly described with means of fracture mechanics. According to fracture mechanics, there are three different failure modes that may occur (or combinations of them). The three failure modes are illustrated in Figure 5-24 below, and they are;

- Mode I – tensile (opening)
- Mode II – shear (sliding)
- Mode III – tear

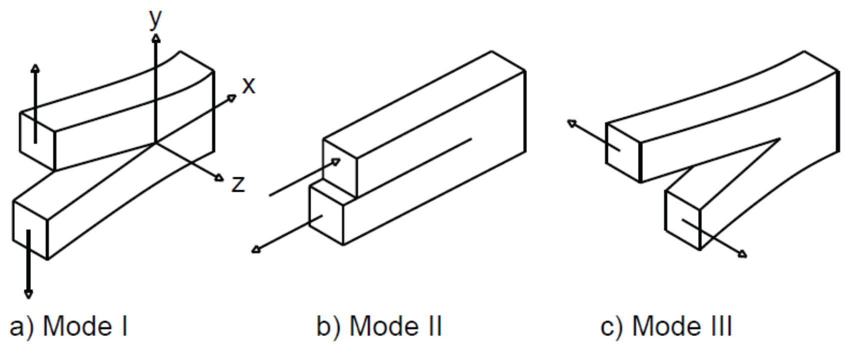


Figure 5-24: Failure modes according to fracture mechanics. Based on Irwin (1958).

In concrete, it is only the mode I failure mode that occur in its pure form. Even for shear type cracks in concrete structures, the crack is initiated as a mode I (i.e. tensile) crack when the maximum principal stress is equal to the tensile strength of concrete. However, in discrete crack approaches the corresponding uniaxial behaviour as described in Section 5.1.2 should be defined for the three modes.

The stress distribution near a crack tip is illustrated in Figure 5-25, for a mode I type of crack. In the figure, a macro-crack has propagated with a length a_0, the fracture process zone is illustrated in the figure as the length l_p. The crack width is denoted as w, and the crack width when the macro-crack just have been developed is denoted as w_c. As it can be seen in the figure, the stress $\sigma(w)$ at the transition between the macro-crack and the fracture process zone is equal to zero, the stress increases in the fracture process zone and is equal to the tensile strength f_t at the crack tip of the micro-crack.

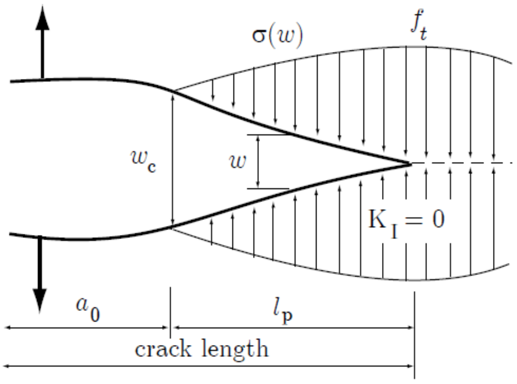


Figure 5-25: Failure modes according to fracture mechanics, from Reproduction from Hillerborg et al. (1976).

The discrete crack approach is defined with a traction-separation law for the interface elements as follows

$$\Delta \boldsymbol{t}^{cr} = \mathbf{D}^{s} \Delta \boldsymbol{w}^{cr}$$

where

$\Delta \boldsymbol{t}^{cr}$ is the traction (tensile stress) vector for the crack

$\mathbf{D}^{s}$ is the secant stiffness for the crack

$\Delta \boldsymbol{w}^{cr}$ is the displacement (crack opening) vector for the crack

One simple case which is commonly used is that the normal and shear stress can be defined to be dependent on the normal and shear displacement only

$$\Delta \boldsymbol{t}^{cr} = (\Delta t_n{}^{cr}, \Delta t_t{}^{cr})^{\mathrm{T}}$$

and

$$\Delta \boldsymbol{w}^{cr} = (\Delta w_n{}^{cr}, \Delta w_t{}^{cr})^{\mathrm{T}}$$

where the secant modulus can be defined as follows for a model based on the fixed crack approach

$$\mathbf{D}^{s} = \begin{bmatrix} K(w) & 0 \\ 0 & \beta(w)G \end{bmatrix}$$

The term $K(w)$ describes the tension softening in the normal direction for successively increased crack opening displacement, starting from crack initiation with micro cracks and to the development of a macro crack, as illustrated in Figure 5-19 a). Different types of crack opening laws (tensile softening curves) where shown previously in Section 5.1.2. The term $\beta(w)G$ describes the shear resistance, which is reduced for increasing cracking opening displacement, as illustrated in Figure 5-19 b).

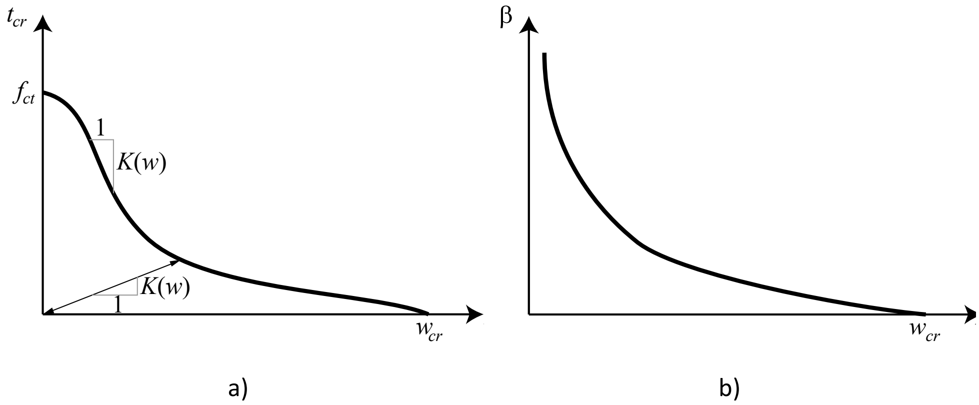


Figure 5-26 Unaxial tensile stress – crack opening displacement curve, b) Shear retention factor.

6 Boundary conditions and interactions

In this chapter, the conventional types of boundary conditions and types of interactions/constraints that may be needed in a typical concrete dam model are described.

6.1 MODELLING THE ROCK MASS

It is rather complicated to accurately modelling the geotechnical aspects of concrete dams. One of the most important aspects of modelling rock mass is to identify potential failure modes in the foundation.

In general when the rock mass is considered in FE-analyses aimed to analyse the structural response of a concrete dam, it is considered sufficient to treat the rock as a continuum and possibly with inclusion of large rock fractures that can serve as potential failure planes, or rock wedges that may influence the dam safety.

Below three levels of modelling detail regarding the rock mass are described.

6.1.1 Boundary conditions defined directly on the dam foundation

In many fields of civil and structural engineering, such as houses, bridges, nuclear industry, it is common to disregard of the underlying rock foundation and to consider it as a fixed boundary condition.

By defining boundary conditions directly on the dam, this leads to a stiffer behaviour. This could be a conservative approach when for instance calculating stresses in the dam. However, these stresses will be unrealistic and may be too conservative in some cases. Typically, significant tensile stresses will occur at the upstream toe due to this boundary condition.

One further downside with this type of boundary condition is that it is not possible to study the effect of uplift pressure if the base surface is defined with boundary condition. Any forces that are directly defined on the nodes that are subjected to a boundary condition will not affect the structure. This may be compensated for, by neglecting a vertical boundary condition on the nodes within the area that is subjected to the uplift pressure. However, this may change the structural behaviour of the dam and should thereby be used with care.

Thereby, this modelling approach is seldom used in dam engineering.

6.1.2 Rock is implicitly considered through springs etc.

The second approach is to attach springs, or other type of special purpose elements to all the nodes of the dam foundation. These springs (or equivalent) can be defined to represent the stiffness of the rock where the spring stiffness for one spring (k) can be calculated as

$$k = \frac{EA}{L}$$

where,

k is the spring stiffness [N/m]

E is the elastic modulus of the rock [Pa]

A is the contributing area of each node [m^2]

L is the height (or length) of the rock mass that the spring is representing [m]. This is further described in Section 6.1.3.

With this approach is easy to defined nonlinear properties of the springs and thereby effects such as sliding or overturning can be simulated. Overturning can for instance be simulated if the vertical springs are defined to only transmit compressive forces (i.e. forces directed downwards) and for tensile forces (upward forces) the spring stiffness is defined to be zero. This is further described in Section 12.2.

Different special purpose elements occurs in different FE codes, which gives the FE-analysist the possibility to define friction in the horizontal direct, where the force in the horizontal element is dependent on the force in the vertical direction.

An example of a case where nonlinear springs and special purpose elements are used to represent the concrete dam can be seen in Figure 6-1.

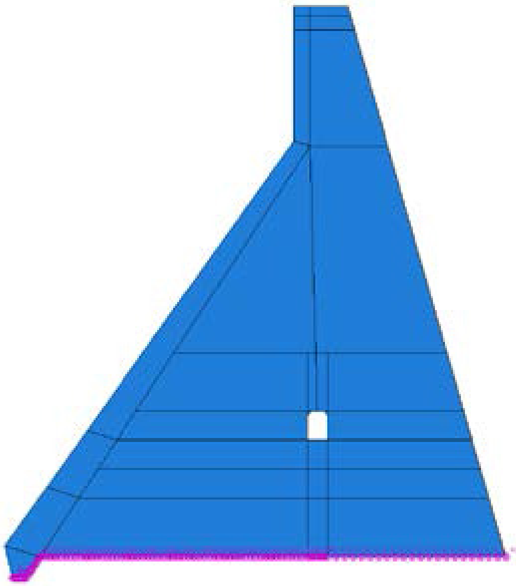


Figure 6-1 Nonlinear springs, representing the rock stiffness, are defined at the base of a concrete dam.

6.1.3 The rock mass is directly included in the numerical model

In the third approach, the rock is directly included in the numerical model. Below some general rules of thumb that can be found in the literature are given

- A general rule of thumb that many use when defining their models is that the amount of rock beneath the dam that should be included in the analysis should be at least equal to the height of the dam.
- The length of the rock mass on the left and right sides (perpendicular to the stream direction) is normally defined equal to the length of the dam.
- In seismic analyses where the water is included, the length of the reservoir should be at least larger than twice the dam height, in order to minimize wave reflection on the outer edge, see Section 8.5. Thereby, the length of the rock is often defined to be as long as the reservoir. It is however common that the reservoir is defined to be longer than the rock mass. This is possible if it is assumed that no waves are transferred from the reservoir into the rock.

If the rock mass is defined with the same size of elements as the dam, then this would result in huge models where the majority of the degrees of freedom are within the rock mass, i.e. in a part of the model where we are not really interested of the results. Thereby, a biased seeding of edges is recommended be used in the model where larger elements can be used in the rock further away from the dam. In addition, most software can (with reasonable good accuracy) constrain two surfaces with large differences in discretization. Thereby it possible to mesh the rock foundation with coarser mesh at the interface with the concrete dam. This is for instance illustrated in Figure 6-2.

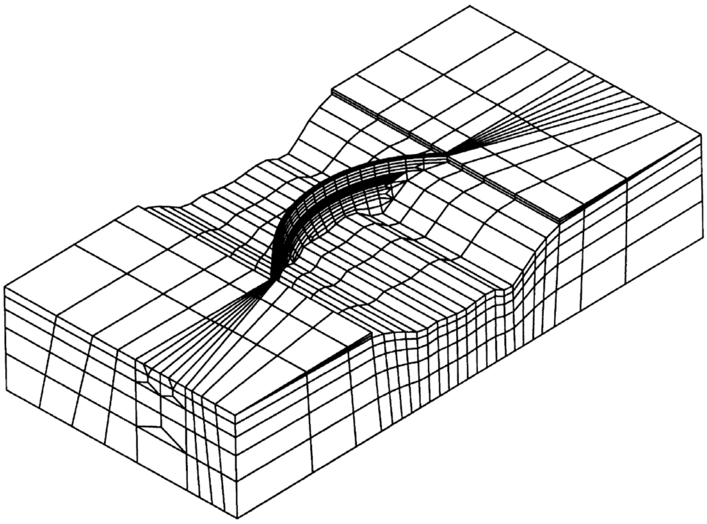


Figure 6-2 Example of biased meshing of the rock, from Malla and Wieland (1999).

The boundary condition is defined on the outer sides of the rock mass, where the rock is constrained to displacements perpendicular to its surfaces, as illustrated in Figure 6-3.

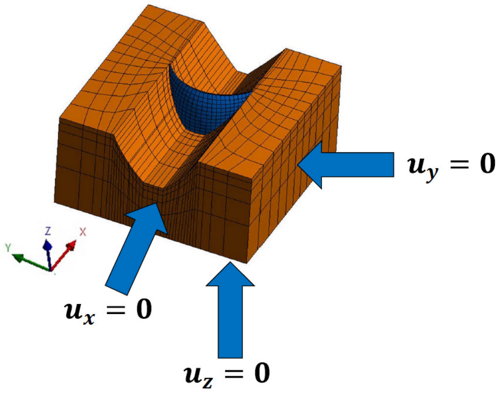


Figure 6-3 Applying boundary conditions to the rock mass, from Shahriari S (2013).

It is also possible for some types of analyses, mainly static analyses, to only include a small part of the rock closest to the dam and to replace the remaining parts of the surrounding rock with springs, as seen in Figure 6-4.

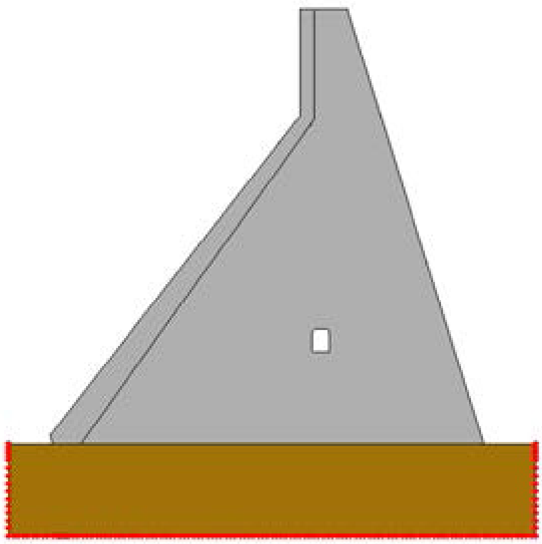


Figure 6-4 Example of a model where parts of the rock is included in the model and the remaining part of the rock is modelled as springs.

6.2 JOINTS AND INTERACTIONS

In all concrete dams, the behaviour of joints and the interaction with the surrounding may have a significant impact on the results. The inclusion of joints within the dam body are important and may influence the response of the dam, depending on how these joints were constructed.

In this chapter, different types of joints that normally occurs on concrete dams are presented and how these could be interpreted in numerical models. The different types of joints considered in the following sections are illustrated in Figure 6-5, and are as follows

- Foundation contact – the concrete is cast on irregular shaped rock often with significant amount of rock bolts. Interaction between dam and foundation is described in Section 6.2.1.
- Contraction joints - are designed to allow for relative movement between the different monoliths, blocks etc. due to shrinkage or thermal expansion. Modelling of contraction joints is described in Section 6.2.2.
- Construction joints (lift joints) - are normally designed to be load carrying, i.e. reinforcement bars crossing the joints. How to consider construction joints in the model is further described in Section 6.2.3

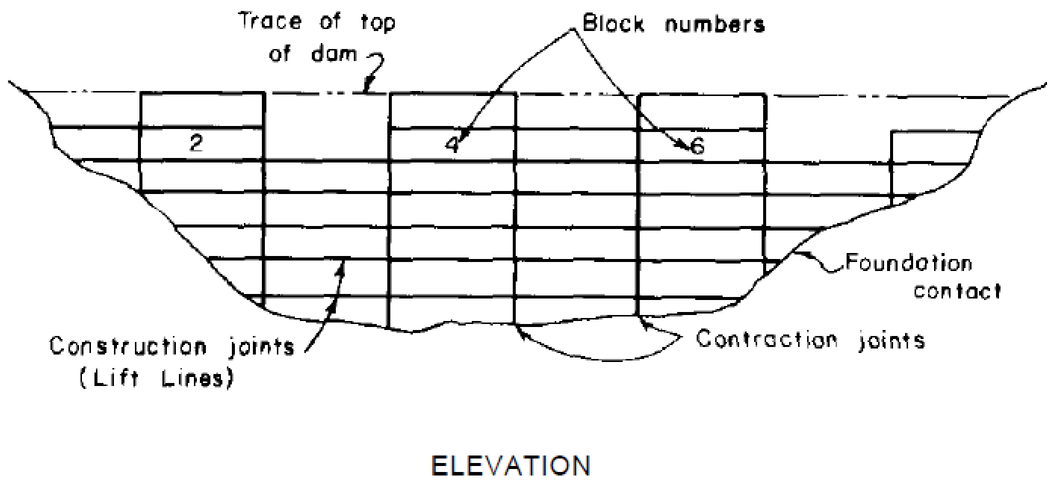


Figure 6-5 Illustration of joints in a concrete dam.

Another type of joint that can be important to consider is existing cracks on old/older dams. These existing cracks can also form potential failure planes or the existing cracks may propagate further due to external loads, which cause potential failure planes. Existing cracks can be modelled in the same manner as contraction joints in concrete. If the dam is reinforced it is important to also include the bars that crosses the cracks in order to also include the resistance of the reinforcement. Modelling of existing cracks is described in Section 6.2.4.

Other types of joints that could be important to consider are large fractures within the rock. This especially, if these fractures can form potential sliding surfaces. Fractures in the rock may be modelled in the same manner as contraction joints or existing cracks, but with a coefficient of friction, that is representative for the specific rock type at the location. Modelling of fractures in rock is described in Section 6.2.5.

Joints, cracks or fractures may play a significant role in the behaviour of the dam under serviceability conditions (i.e. for normal loads). However, as the intensity of loads increases and becomes close to the capacity of the dam the larger impact these will have. In general, for concrete dams, nonlinearities due to existing cracks and contraction joints are usually activated first as soon as the loads are sufficient to mobilize the friction in the cracks or to open these cracks. Nonlinear behaviour of the contraction joints is activated as soon as the load intensity is sufficient to break the bond in these joints. As the load intensity continues to increase, material nonlinearity such as forming of new cracks or yielding of reinforcement in existing cracks may also occur. Finally, as the strains and displacements are further increased, the general assumption of small displacements may no longer be valid and hence second order effects (geometric nonlinearity) also may have to be considered. These second order effects are however, rarely of great importance and can for most cases be neglected. (ICOLD, 201x)

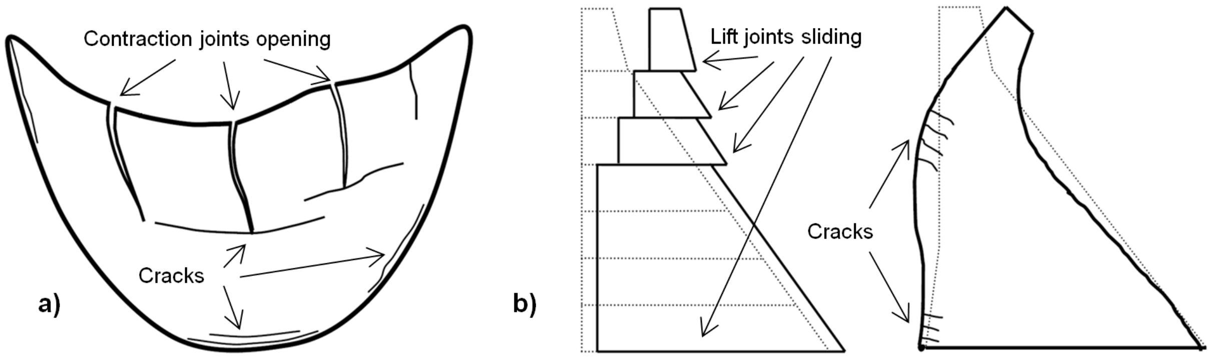


Figure 6-6 Example of failure modes due to joints or cracks in the dam body, from ICOLD (201x).

6.2.1 Interaction Concrete and foundation

As described in Section 6.1, in most cases, part of the rock foundation will be included directly in the numerical analysis. There are different methods on how this interaction can be done and it depends on the problem that is going to be solved.

One common assumption is that the dam is assumed to be completely bonded to the rock foundation. This can in some cases be a fairly accurate assumption because the concrete will have some bond strength to the rock surface, especially if the rock surface is irregular (blasted) and was cleaned and free of loose particles/dirt. The bond (cohesion) and shear strength is however, in general lower than the tensile and shear strength of concrete and in these cases. Hence, it should be ensured that not high tensile stresses occur in the dam body closest to the rock. For seismic analyses, assuming complete bond between concrete and rock is the most common alternative. However, using this approach, unreasonable high stresses may be found at local points near the foundation, especially in the upstream toe of the dam. According to FERC (1999) high tensile stresses may be acceptable if they not indicate the actual stress-state but instead is a result of simplifications or assumptions made in the model. This typically can occur near the upstream toe of the concrete dam. Complete bond may also induce large cracks or high tensile stresses in the concrete dam when shrinkage and/or temperatures are considered.

A more realistic modelling assumption is to define the contact between the dam and the rock foundation with contact surfaces, interactions, nonlinear springs etc. With these types of connections it is possible to include cohesion if deemed necessary[2] but also include friction in the horizontal plane and a contact law that allows for the dam to release from the rock if subjected to tensile forces, for instance due to an overturning failure. Below, an example of an interaction between concrete and rock is presented that was used by Goldgruber (2015).

As a general recommendation for modelling contact between two bodies where the element size may differ, is that the part with coarser mesh should be defined as the master surface. This results in general in fewer problems with lack finding convergence. The reason for this is that the mid-nodes on the slave surface with finer mesh will be adjusted to remove penetration based on a linear interpolation between the nodes of the coarser master surface. If it is defined the other way around, there will several master nodes that connect to the same slave nodes and thereby several additional iterations may be required to find equilibrium.

One additional recommendation is to make the contact surface of the master (the rock foundation) larger than the contact surface of the slave (i.e. the dam), i.e. extend the master surface so it is larger than the footprint of the dam. The reason for this is that the dam will slide away, and if the master surface is larger, then it is certain that all nodes will be able to find contact even though some deformation has occurred. An example of this is shown in Figure 6-7.

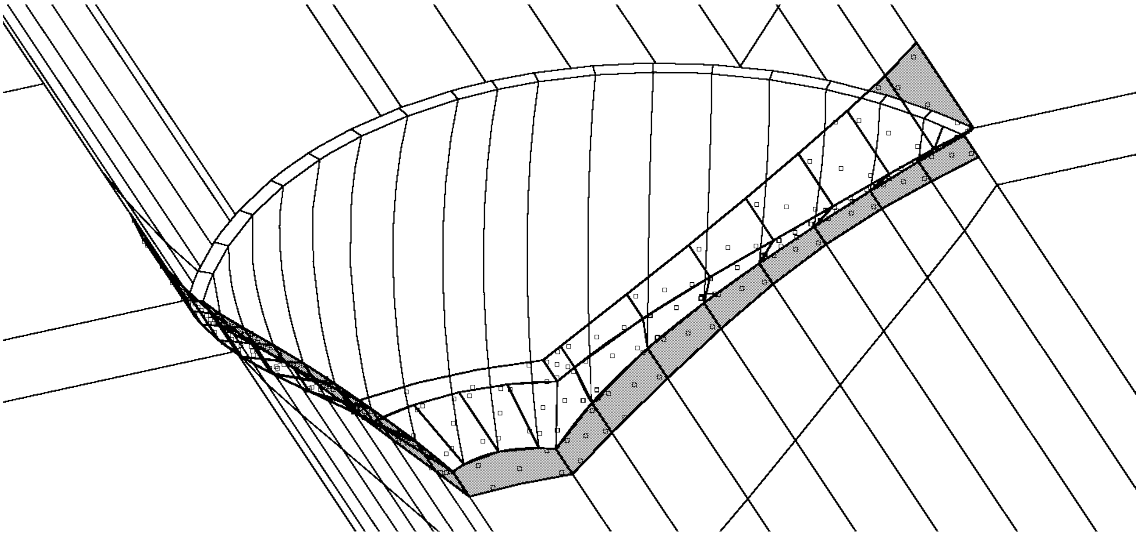


Figure 6-7: Extension of the master surface (foundation), highlighted in grey, from Goldgruber (2015).

One additional aspect that is important is that not to lose tolerances are defined for the contact slip tolerance. As for all tolerances, it is (in general) better to define rather strict tolerances and allow for more iterations than the opposite.

The coefficient of friction is in general given in guidelines and standards regarding the concrete dam and the foundation, depending on different ground materials. As

[2] It should be noted however, that it is not allowed to include cohesion according to the Swedish guideline RIDAS (2011).

an example, it can be mentioned that the coefficient of friction for concrete on rock of good quality can be estimated as μ=1.0 according to RIDAS (2011)[3].

The contact in the normal direction should be defined to allow for compressive stresses only, i.e. occurrence of tensile stresses should lead to separation of the dam and the foundation at this point. For practical reasons it can in some cases, for instance with nonlinear springs, improve convergence for initial steps if some negligible amount of tensile forces are allowed to be transmitted, for instance as illustrated in Figure 6-8.

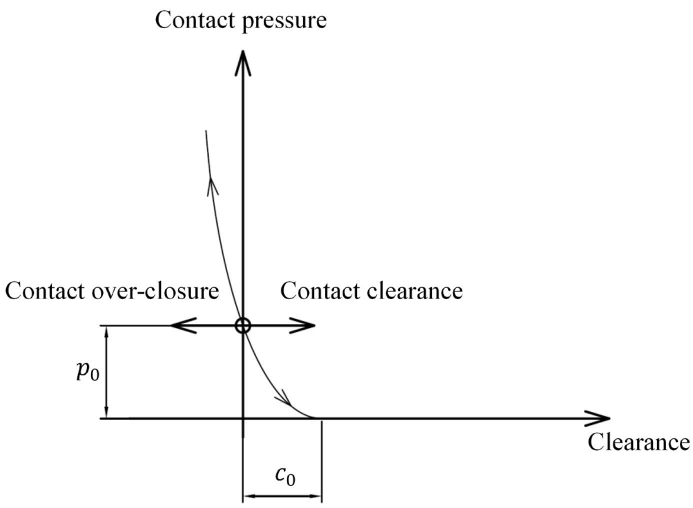


Figure 6-8 Example of a "soft" contact, which may help improve convergence, reproduced from Dassault (2014).

6.2.2 Contraction joints

Contraction joints may used to limit the size of the numerical model, since many joints are not constructed to transmit any forces and hence it is convenient to define the numerical model so that its boundaries are located at these joints. This was previously described in Section 3.2.

Many dams are constructed where the contraction joints are formed as shear keys and thereby resulting in high shear resistance in the stream direction but no (or very small) tensile, opening, forces and low shear resistance in the vertical direction. These kinds of construction joints are typically found on large arch dams, as seen in Figure 6-9.

[3] A safety factor of s_g = 1.35 is then required for sliding failure under normal loads

Figure 6-9 Construction of an arch dam where the shear keys of the contraction joints are visible, from James and Dollar (2003).

The coefficient of friction may vary significantly in contraction joints depending on its design. In general, the coefficient of friction may vary between μ=0.3 (smooth surfaces) up to μ= 2.0 (joints with large shear keys).

In the paper of Tassios and Vintzēleou (1987), the load transfer along unreinforced concrete interfaces for smooth, rough and sandblasted surfaces was investigated. They found that compressive stresses between 0.5 – 2.0 MPa across the sliding plane had minor impact on smooth surfaces and resulted in a coefficient of friction between 0.4 and 0.5. The case with rough sliding surface showed larger influences of compressive stresses. Therefore, cyclic tests were performed to study the development of the coefficient of friction as the rough surfaces are grinded against each other. These tests lead to values between 0.5 and 0.7 for relatively small displacements after the sixth cycle for a compressive stress of 0.5 to 2.0 MPa.

Goldgruber (2015) showed a case where different approaches where used to calculate the behaviour of an arch dam depending on different approaches to consider the contraction joints. Four different approaches are showed in Figure 6-10. In all figures, an illustration of the vertical stresses in the dam is show for a gravity load step.

a) The dam is considered as an isotropic, monolithic, structure, i.e. no consideration of contraction joints.
b) The dam is defined with an orthotropic material definition for the tangential, radial and vertical direction.
c) The cantilevers are modelled as separate columns/blocks without interaction between each other.

d) The cantilevers are modelled as separate columns with interaction describing friction and hard contact that allows for separation in the normal direction. (This is the most correct approach to use)

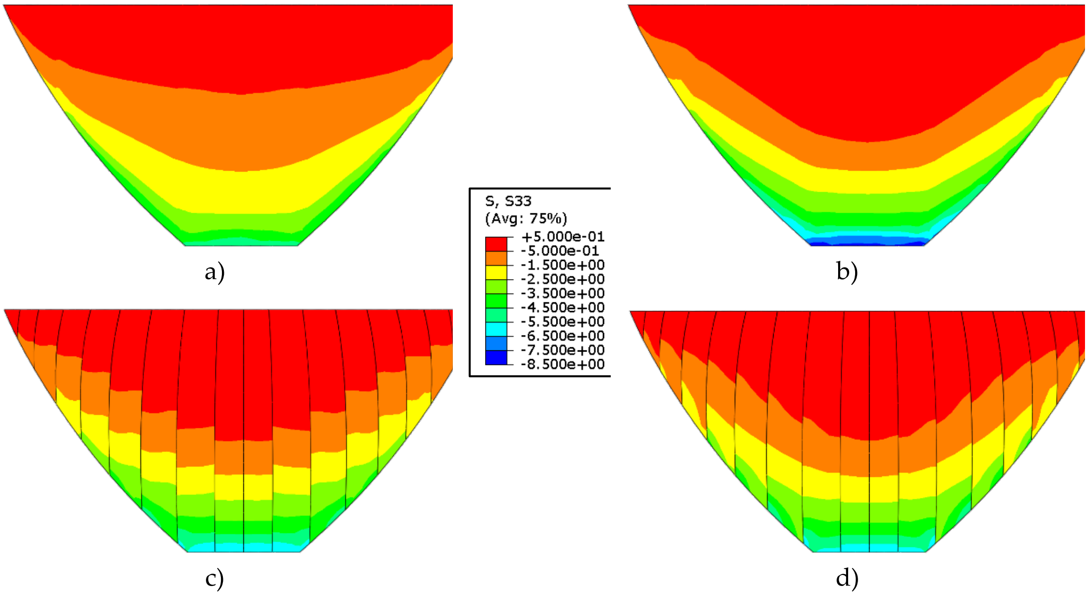


Figure 6-10: Qualitative vertical stress comparison for different dead weight modelling techniques: a) Without consideration of separated columns (isotropic); b) Orthotropic material behavior; c) Separation of columns (without interaction); d) Discrete columns with contact. From Goldgruber (2015).

The first approach, a), with an isotropic model is the most common approach found in the literature. Goldgruber (2015), concludes that this rough approach leads to the wrong behaviour of the whole dam. The reason for this is that shear stresses between columns are fully transferred and the columns are thereby "hanging" on each other. Therefore, an isotropic model of the dam is not recommended, since it may over- or underestimated the stresses in the structure.

One other common approach is approach c), where the cantilevers are modelled as separate columns. In this approach, it is common to separate the simulation of the gravity load in three steps. In these steps the gravity load on different block sets are calculated. In the first step, e.g. every block with an even number and in the second step every block with an odd number is simulated. In the third step, the results from the blocks are superposed, giving the stress level in the structure from the gravity load for subsequent steps. This approach shows a better stress distribution, and hence describes the structure in a more correct manner. However, this approach is only valid to a linear system (i.e. where it is possible to use superposition of the results). The results shows that calculated stresses indicates discontinuities between the columns, and hence is therefore over- or underestimating the stresses in some of the columns. Despite this, the method gives relatively good results.

The principle with the orthotropic material model is that only properties that should be considered in the different directions are given relevant values, while other effects are defined to be equal to 1 % of the correct value.

- The orthotropic material model is defined where the radial and vertical direction are defined with the correct elastic modulus while the tangential direction is defined with only 1 % of the elastic modulus.
- Poisson's ratio is defined its correct value only in tangential direction and 1 % of this value in the other directions.
- The shear modulus is defined with its correct value only in the tangential direction and with a reduced value corresponding to 1 % at the other directions.

This approach shows that an orthotropic model gives an overall good stress distribution that corresponds well with the more detailed model in Figure 6-10 d). One difference is however, that the approach leads to higher stresses near the upstream toe.

The approach shown in Figure 6-10 d), with discrete columns with contact between the columns is the most accurate approach of these four and is therefore recommended to use.

6.2.3 Construction joints (Lift joints)

In cases where the construction joints are designed to be load carrying, i.e. reinforcement bars crossing the joints, it is usually sufficient to consider the dam (or dam monolith) as a monolithic structure.

If site-investigations have shown that the dam is expected to be weakened by these lift joints, then it is suitable to also consider these in the model. This can be performed in several different ways, for instance

- Element rows representing the area closest to the lift joint are defined with reduced concrete strength (chosen based on measurements or for instance as lowest properties according to Eurocode i.e. C12/15)
- Define discrete cracks at the region of the lift joints and use a contact formulation to describe, cohesion, friction etc.

The lift joints are possible failure planes, since the bond strength between the different lifts is expected to be less than the strength of the concrete. Hence, during the evaluation of a concrete dam, extra care should be taken if large tensile stresses occurs within the dam body in an analysis where the dam is described as a monolithic structure. It is especially sliding failure within these lift joints that should be evaluated and it should also be considered that an uplift pressure may occur in these lift joints if tensile stresses occurs.

In the numerical software CADAM, it is possible to perform stability analyses where the lift joints are considered and also potential uplift pressures in these joints.

6.2.4 Existing cracks

Existing cracks in the dam body may have significant influence on the behaviour of the dam. It is therefore important to include the effect of significant cracks in the model, if for instance the safety of the dam is to be estimated or if strengthening solutions are being designed/planned.

An existing crack in a dam can be modelled in the same manner as using the discrete crack approach. This may for instance be defined by assembling a model consisting of two parts on each side of the crack. Between the two geometric bodies, a contact formulation may be used to define properties for the macro crack and if parts of this crack is to be considered as intact concrete, then another contact formulation is required for the part representing intact concrete. If the concrete dam is reinforced with rebars, then it is important to define truss elements or reinforcement layer elements that represent the reinforcement that can transmit tensile forces across the crack plane. An example of a study where existing cracks have been included in the model is shown in Figure 6-11.

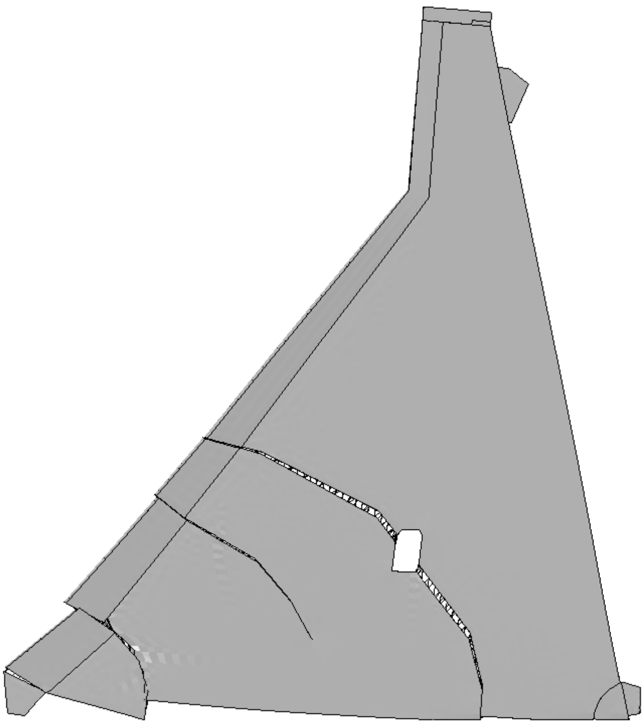


Figure 6-11 Example of a 3D model of a buttress dam with pre-defined existing cracks. (The lines crossing the cracks are truss elements representing reinforcement bars). From Malm et al. (2016a).

An existing macro crack can be defined with properties similar to construction joint, i.e. frictional behaviour in the tangential direction and in the normal direction, only compressive forces should be transmitted.

In some software's, it is possible to define the dam as one geometric body and then define lines or surfaces that represents cracks within the structure. At these lines or surfaces, overlapping nodes are generated so that relative opening can occur within the crack. An example of this approach is illustrated in Figure 6-12.

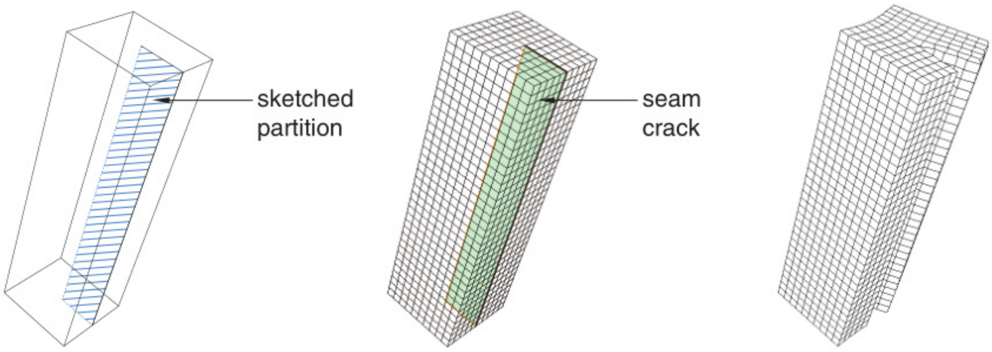


Figure 6-12 An example of a method to define a discrete crack in a geometric body, from Dassault (2014).

6.2.5 Rock fractures

Large fractures within the rock can form potential sliding surfaces and should be considered in the analyses. A failure in the rock will be initialized for the failure mode with the lowest strength. According to Gustafsson et al. (2008), three different failure modes are used to categorize failure in the rock for concrete dams with a rock foundation, as seen in Figure 6-13.

a) Failure in the contact plane between concrete and rock
b) Failure along a fracture plane or other sources of weakness within the foundation
c) Failure in the rock mass

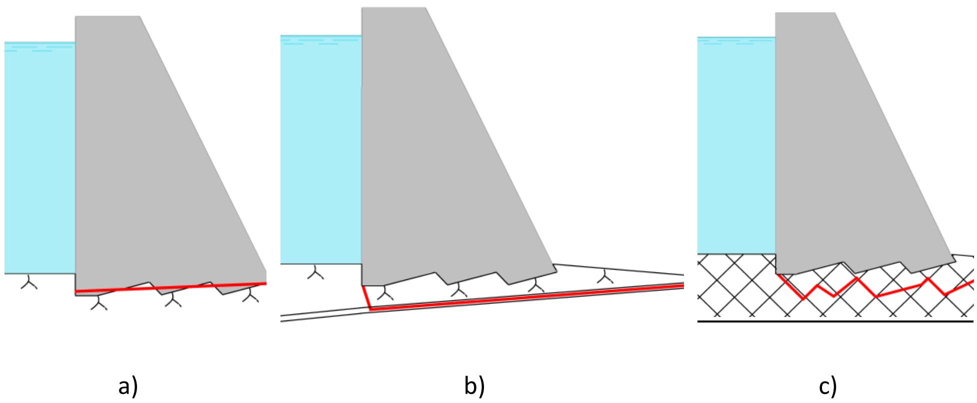


Figure 6-13 Potential failure planes in the rock due to rock fractures, a) failure in the contact plane between rock and concrete, b) failure along a fracture plane or other weaknesses in the rock, c) failure in the rock mass. From Gustafsson et al. (2008).

Fractures in the rock may be modelled with contact formulations with interaction properties with coefficient of friction, possible cohesion and a contact formulation that allows for opening of the rock fracture. Interaction properties are later described in Section 6.2.1.

In cases where modelling fractures in rock it is important to determine a coefficient of friction, that is representative for the specific rock type at the location. In these cases, it is important to consider if the fractures are filled with particles or not since this has a significant influence on its shear resistance. Different models to describe the behaviour of fractures in rock and material properties is for instance summarized in Gustafsson et al. (2008), Johansson (2009), Elsayed (2012), etc.

7 Static analyses

Depending on the application, different types of loads have to be considered. In this section, the most common types of loads acting on dams are described and how these may be defined in a numerical FE model. Depending on the specific project, different types of loads may also need to be considered.

Specific load values are not given in general in this section, these should be chosen according to each specific national standard/guideline. Thereby different load values may be defined in different national guidelines and/or requirements. In Sweden, these load values should be chosen based on the Swedish guideline for dam analyses, RIDAS (2011).

Depending on if, the model is used for global assessment of the dam or used for detailed analysis or design of a local area of the dam, the modelling detail may differ regarding how different loads are modelled.

Analyses performed on dams are performed with continuum elements, i.e. solid elements (in 2D or 3D) or shell elements (3D) depending on the structure, as mentioned in Section 2. Due to this, concentrated forces (point loads) and concentrated boundary conditions (point boundary conditions) should be avoided as far as possible. The reason for this is that a point load/point boundary condition will introduce stresses in the continuum that approaches infinity with reduced element size in linear elastic analyses, as shown previously in Section 4.2.

In analyses based on nonlinear material behaviour, these unreasonable stresses will be distributed in the structure, however localized forces might cause instability and convergence issues that make the analysis much more difficult to perform.

7.1 MECHANICAL LOADS

In this section, the most common types of mechanical loads that has to be considered in numerical analyses of concrete dams are described. The focus of this section is how to include the loads in the numerical models. The actual load value for each load, may be found in design codes/guidelines, such as RIDAS (2011).

Many other type of loads that are not considered in this section may act on concrete dams, such as wind loads, traffic loads, wave loads, explosions etc. Wind and wave loads does however, in general have relatively small impact on the structural response of the dam. The traffic loads may have a significant effect if there is a public road on the dam crest. In many cases, the road on the dam crest is private with only the traffic of vehicles that are going to the hydropower plant. The size of the traffic load may be assess by Eurocode EN 1991-2. An engineering judgement has to be used in each project regarding the type of loads that may affect the dam and that has to be considered in the analyses.

7.1.1 Gravity loads

Gravity loads are in general considered as body forces in the numerical model. Thereby, the density and the gravitational acceleration are given. The gravity forces are thereby calculated based on the volume of the modelled structure

$$F = \sum V \cdot \gamma$$

where,

V is the volume of the structure/structures [m^3]

γ is the weight of the material/materials [kg / m^3]

Gravity loads from objects that are not included in the model should be applied to the dam structure as distributed loads. The easiest way to do this is to apply the load as a pressure. Alternatively, it can also be done by multi-point-constraint (MPC) by applying a point load to a discrete node, which is attached to an area of the structure. In dynamic analyses, the mass of these objects should be included since it affects the dynamic behaviour of the dam.

7.1.2 Hydrostatic water pressure

In many software's there is a possibility to define a pressure load, where its amplitude is dependent on the z-axis in 3D or y-axis in 2D. In these cases, the dam height has to be defined in this direction and the FE-analysist specifies the two altitudes that the pressure should be defined between, i.e. bottom of the reservoir and the water level at the upstream side. The hydrostatic pressure is calculated as:

$$p_i = \rho g h_i$$

where,

p_i is hydrostatic water pressure at a specific depth h_i [Pa]

ρ is the density of water [kg / m^3]

g g is the gravitational constant [m/s^2]

h_i is depth at a specific point i [m]

In those cases where this is not possible, it is usually possible to define that the pressure load is dependent on an analytical function which defines that the amplitude is a function of the dam height.

The hydrostatic pressure from the reservoir should not only be defined on the upstream side of the dam, it should also be defined on the upstream part of the rock. An example of the applied hydrostatic water pressure is shown in Figure 7-1. The same applies to the downstream side if the dam is subjected tailwater.

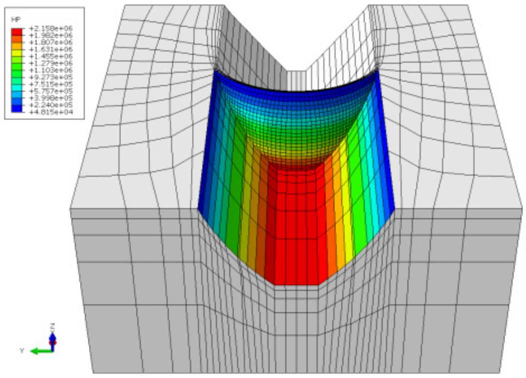


Figure 7-1 Applied hydrostatic pressure on an arch dam.

The hydrostatic water pressure is usually defined normal (usual) level corresponding to highest elevation that the water normally is stored. Exceptional flooding scenarios may result in that the water level exceeds the dam height.

An alternative, and more correct, approach to consider the effect of water pressure, is to simulate pore pressure distributions within the dam and foundation instead of applying the water pressure as an external surface pressure. This is, however, more complicated methodology and requires that a transport analysis is performed initially to calculate the pore pressure distribution. The calculated pore pressure distribution is then used as input in a following mechanical analysis, where the pore pressure can be converted into volumetric stresses in the mechanical analysis. A schematic sketch of required material properties and how the boundary conditions are defined in the pore pressure analysis is shown in Figure 7-2. Advantages with pore pressure analyses is that they at the same time also gives results for uplift pressure and that it is possible to more in detail study the pore pressure distribution due to grouting curtains, drains etc.

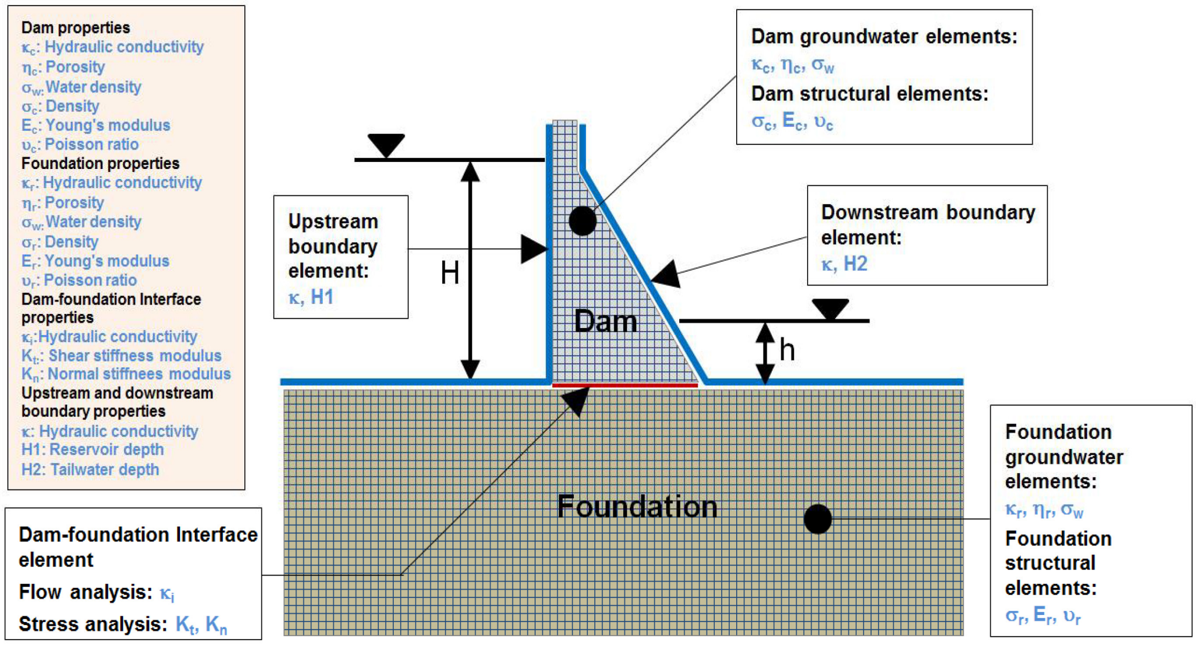


Figure 7-2 Schematic illustration of a pore pressure analysis, from McKay and Lopez (2013).

7.1.3 Uplift pressure

Several different distributions regarding uplift pressure for gravity dams are given in Section 7.3 in RIDAS (2011). These distributions may in the simplest approach be applied directly in a FE-model as pressure distributions with an analytical function describing its distribution in the downstream direction. However, in this case the uplift pressure will not be dependent on joint opening (i.e. a cracked base analysis) between the concrete dam and the rock, which would lead to much higher uplift pressures, move the resultant force downstream and thereby reduce the overall safety of the dam.

In many cases, more detailed analyses may be required. In these cases, linear or nonlinear pore pressure and/or uplift pressure analyses are required. There are according to McKay and Lopez (2013) fairly limited amount of literature that presents how to include uplift pressure in FE-analyses.

FERC (2002) is one of few that presents a simplified approach on how to include the uplift pressure in FE analyses. Their methodology is illustrated in Figure 7-3 below. In this methodology, it is suggested to define thin interface element which allows for uplift pressure to be defined at the base of the dam and at the top surface of the rock mass. The resulting stress output for these interface elements includes the effects of uplift. A cracked base analysis may be performed in an iterative way by deleting elements in the cracked zone and manually recalculating the uplift distribution. According to McKay and Lopez (2013), this method is relatively easy to implement in 2D models but is more impractical for 3D models.

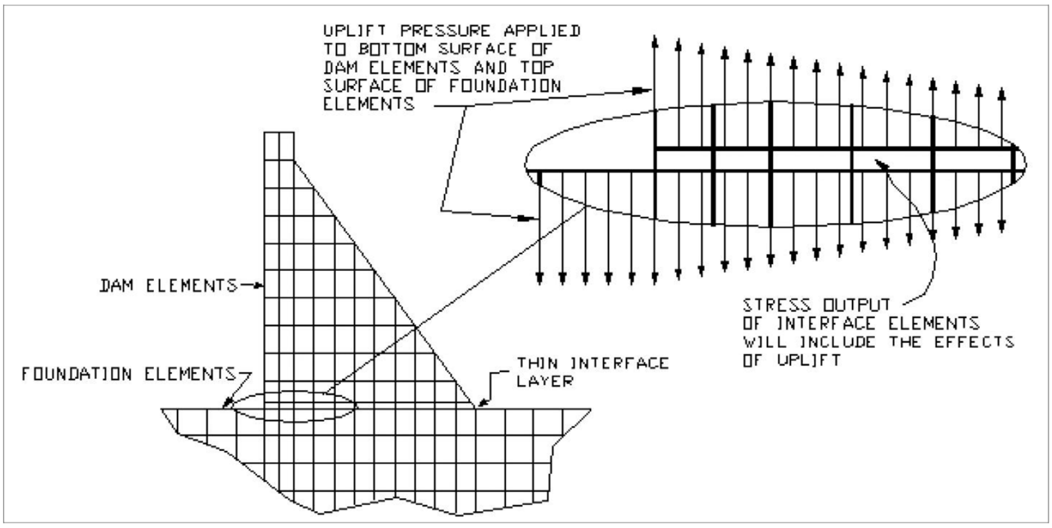


Figure 7-3 Methodology for applying the uplift pressure, from FERC (2002).

Another approach is suggested by USBR (2006), where the uplift pressure should be defined as surface pressures to the opposite faces of contact elements. This method can be implemented in 2D and 3D FE-models but it requires that the analysis is performed with boundary/interaction nonlinearity. Performing analyses with this form of nonlinearity does not, in general, result in problems with convergence issues. It is still not straightforward to implement a successively increasing uplift pressure as the joint (crack) along the base opens up.

McKay and Lopez (2013) have for suggested a simplified approach for pore pressure analyses where sequential temperature and pore-pressure analyses (i.e. transport analyses) assuming steady-state conditions are performed and used as input in a mechanical analyses to calculate the stresses. In the paper of McKay and Lopez (2013), a methodology to iteratively adjust the uplift pressure is presented. However, this approach is based on manual adjustments in the pore pressure analysis by adjusting the pressure heads assigned to each boundary element. Thereby, neither the method USBR (2006) or McKay and Lopez (2013) includes an easy way of automatically iterate the uplift pressure due to propagation of a crack in the base.

7.1.4 Ice loads

In countries with cold climates, ice loads may be significant. The ice loads that may occur could occur due to ice expansion in an enclosed area, for instance between dam pillars or at gates or global ice loads along the whole dam water line. Regardless if localized effects or the global effects from ice loads are considered, the ice load should be defined as a distributed pressure load.

National requirements or guidelines defines the size of the ice load and its assumed thickness. As an example, the ice load in the northern part of Sweden should be assumed to be 200 kN/m along the dam water line and the ice thickness should be assumed as 1 m according to RIDAS (2011). In RIDAS it is defined that the force resultant of the ice pressure should be considered to be located at the distance $h_{ice}/3$ from the top of the ice sheet. Thereby, in order to define the ice load in conjunction with RIDAS (2011) it should be defined as a triangular pressure where the maximum pressure is equal to

$$p_{ice,max} = \frac{2P_{ice}}{h_{ice}}$$

where,

$p_{ice,max}$ is the ice pressure at the top of the ice sheet [Pa]

P_{ice} is the design load for ice pressure [kN/m]

h_{ice} is the thickness of the ice sheet [m]

7.1.5 Earth and silt loads

It is common that parts of the dams are in contact with soil; this could for instance be due to soil backfill or in a case where a concrete dam is connected to an earthfill dam and the soil is therefore wrapped around the closest monolith/monoliths. The fill material may or may not be submerged.

- Silt pressures should be considered in the analyses if suspended sediment measurements indicate that such pressures are expected.
- The horizontal soil pressure depends on the lateral deformation of the structure, where it can be defined as active, passive or at-rest.

The size of the silt pressure and/or earth pressures are given in design codes/guidelines such as in Annex C in Eurocode 7 (2010) regarding static pressures or for dynamic analyses according to Annex E in Eurocode 8-5 (2009) or for instance in USACE (1989) or Mikola and Sitar (2013).

Design values for earth pressures are given RIDAS (2011), and in case of more appropriate values are available the properties given in Table 7-1 may be assumed.

Table 7-1 Soil properties for determination of earth pressures.

Material	Weight [kN/m^3]		Friction angle [°]
	a.s.l.	b.s.l.	
Excavated rock	17.5	11	42
Gravel	18	11	35
Sand	18	11	32
Moraine	21	13	34

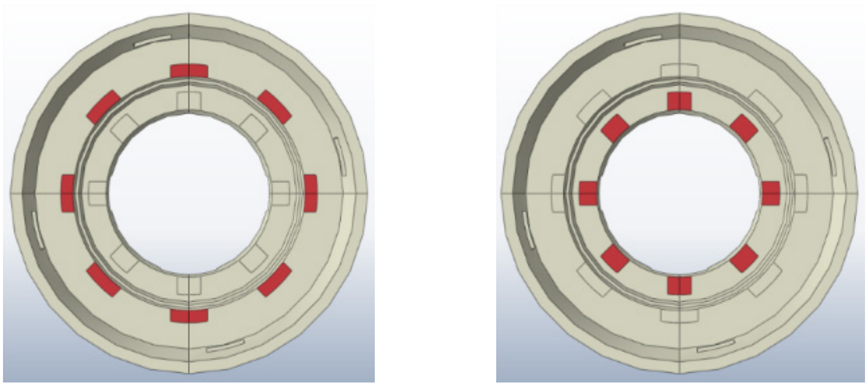
In addition, it is defined in RIDAS (2011) that silt pressures does not have to be considered for Swedish dams.

In static analyses, the soil may be assumed as a distributed pressure loads in the FE-model.

7.1.6 Concentrated forces (from gates etc.)

As mentioned previously, concentrated forces, i.e. point loads, should be avoided as far as possible when working with continuum based models. These concentrated forces should instead be distributed over a surface, where the area of the surface should correspond to the load distribution at the actual structure.

One example of this could be when analysing the concrete structure in the powerhouse that constitutes a foundation for the generator. In these cases, the forces from the generator are transmitted through steel beams to bearings. In Figure 7-4, an example is shown where the forces from the generator are applied as surface loads at the stator and rotor bearing supports.



a) b)

Figure 7-4 Surfaces corresponding to the bearings of stator (a) and rotor spider (b) supports. From Malm et al. (2013b).

In more complex cases, or when higher level of detail is required, it is possible to define some of the structural parts as "modelling aid". This means that the modelling aid structures are only included in the numerical model to distribute forces etc. and the actual behaviour of these parts is not of interest.

As an example, this can for instance be a case with a global analysis where it is important to consider the forces from the gates that are transmitted to the concrete structure. In this case, a pressure load is on the gate that consists of a rigid body shell element structure. The forces are then transferred from the gate through a MPC (multi-point-constraint) to a reference node. The force in the defined reference node is then distributed over a surface in the concrete structure, as shown in Figure 7-5.

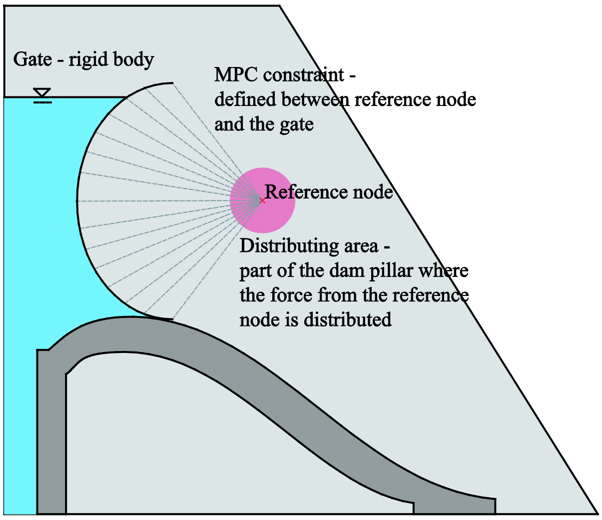


Figure 7-5 Example of a model where the loads on the gate are transferred to the concrete.

In more detailed analyses, where it is important to analyse local effects around the gates, it might be necessary to define detailed 3D models of the actual design of the gate attachment, as illustrated in Figure 7-6.

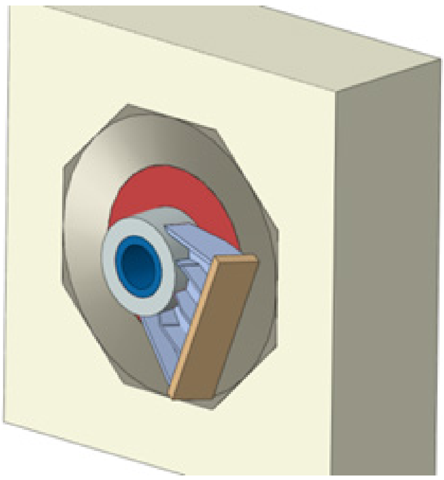


Figure 7-6 Example of a gate attachment.

7.1.7 Pre-stressing tendons

Post-tensioned cables are rather common in concrete dams, and are mainly used as a measure to increase the dam stability. The advantage with pre-stressed tendons compared to rock bolts is that the tendons are active and contributes with a restraining force without the need for deformations. The rock bolts requires deformations before they start to carry forces.

There are many different levels of detail that can be used depending on what is required for the specific project/case. The main distinction between tendons is that it can be grouted (with cement) or unbonded (i.e. in grease). Grouted tendons are in general a bit easier to analyse while unbonded tendons require more effort to include friction along the tendon etc. Suggestions on how to model these cases are described in Table 7-2.

Table 7-2 Level of detail regarding modelling of pre-stressing tendons.

	Tendon forces	Interaction with surrounding concrete
Grouted tendons – minimum approach	Constant initial stress or equivalent temperature along the tendon (tendon stress should be held constant within the first equilibrium iterations)	Nodes of the tendons are constrained to the surrounding concrete.
Grouted tendons – recommended approach	Variable* initial stress or equivalent temperature along the tendon** (tendon stress should be held constant within the first equilibrium iterations)	Nodes of the tendons are constrained to the surrounding concrete.
Unbonded tendon – minimum approach	External compressive force at the anchor.	No cable is needed.
Unbonded tendon – recommended approach	Variable* or constant initial stress or equivalent temperature** along the tendon (tendon stress should be held constant within the first equilibrium iterations)	Constrained that allows for tendon sliding along the duct. This can be done in several different ways, as described in Section 12.6.
Unbonded tendon – detailed approach	Pre-stress and seating is prescribed with special-purpose elements and the variation in pre-stress along the tendon is calculated based on friction. This is further described in Section 12.6.	Constrained that allows for tendon sliding with friction along the duct, see Section 12.6.

*Variable pre-stress along a tendon may be calculated based on analytical calculations considering the losses and applied as an initial stress in the FE model.

** One alternative approach could be to define a temperature distribution along the tendon and based on an expansion coefficient the temperature is converted into strain/stress.

7.2 ANALYSES OF CONSTRUCTION PROCESS

Concrete dams are, like most other types of concrete structures, built by casting segments. The idea when planning the casting of these segments is to reduce internal stresses due to hydration and shrinkage between segments of different age. Concrete gravity and arch dams are mainly built with the high-low block methods, as seen in Figure 7-7. The contraction joints between each column is grouted typically in 18 meter (60-foot) high intervals, according to USBR (2006). This means that the blocks are not in contact until they are grouted.

Figure 7-7 High-low block method used to build Zillergründl arch dam.

In order to calculate the gravity load accurately and the initial condition of a dam, the construction process has to be considered where the staged construction and grouting process are modelled.

A common, and simple approach to simulate this in FE-analyses is to separate the simulation of the gravity load in three steps. The cantilevers are numbered consecutively, and in these steps the gravity load on different cantilevers are calculated. In the first step, e.g. every cantilever with an even number is analysed and in the second step the cantilevers with an odd number is simulated. In the third step, the results from the blocks are superimposed, giving the stress level in the structure from the gravity load for subsequent steps. It should be noted that this procedure is only applicable to a linear system behaviour, because of the superposition of the results and it does not account for interaction between the blocks. (Goldgruber, 2015)

An alternative and more detailed approach to simulate the construction sequence is to simulate the casting of segments/blocks through the use of a "birth" commando which is available in most FE-codes. In this approach, it is possible to introduce new elements successively in the model throughout the analysis of the construction. Thereby it is possible to perform the staged construction. In addition, most FE-codes also makes it possible to introduce interactions in the same manner, which can be used to simulate the grouting of the contraction joints. (USBR, 2006)

An example of this is shown in Figure 7-8. When simulating a staged construction it is important to also include the development of material properties for young concrete. This is described further in Section 7.2.2.

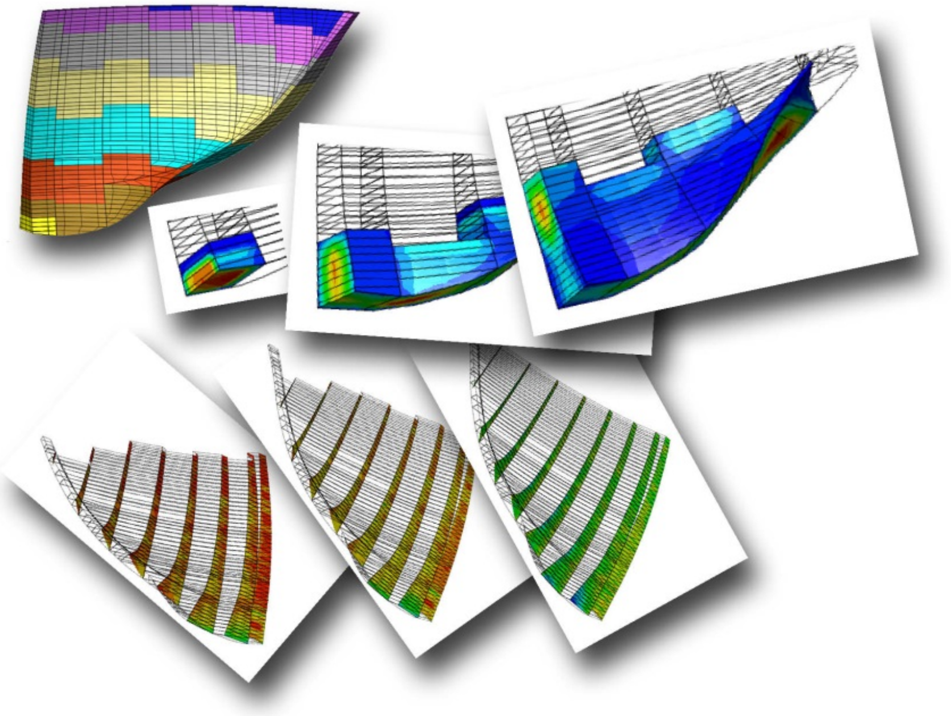


Figure 7-8 Example of a simulation of staged construction, from http://tnodiana.com/.

7.2.1 Disregarding of displacements from the dead weight modelling

In mechanical finite element analyses, a system of equations are solved where the applied loads causes displacements and hence strains and stresses. This implies that a structure needs to deform in order to be subjected to stresses. When analysing the response due to gravity loads, this is not the case since displacements during construction are compensated in order to build the according to the design drawings. As long as these displacements are small and if a linear analysis is performed, then this effect can be ignored and the static deformations from gravity load can be subtracted from the final results caused by other loads. In nonlinear analyses, this might not be a valid procedure, since stresses and displacements are dependent on their history. Thereby, in nonlinear analyses it might be required that the analysis with gravity load is performed in a manner where only the stresses from the gravity load step are included in subsequent steps. (Goldgruber, 2015)

This can be performed in different ways in a finite element analysis. One solution is to perform the gravity load step as a separate analysis and the obtained stress distribution is used as initial stresses in a second model that considers the subsequent load steps. In some FE-software's specific analysis procedures intended for this purpose are available, such as for instance the Geostatic step in Abaqus.

7.2.2 Including hydration heat and strength growth of concrete

The approach mentioned earlier in this section is often used to simulate the influence of the gravity load. In some cases, it important to analyse the construction sequence and thereby to include the development of material properties with time and the heat generated during hydration.

Simulations of hydration heat are normally performed in transient thermal analyses, where the generated heat is defined as dependent on time and is included in the models as new casting segments are introduced. One method to do this is for instance described in Blomdahl et al (2016). The results from this transient temperature analysis is then used as input in a following mechanical analysis.

In most FE-software's it is possible to define that the material properties are dependent on time and other quantities such as temperatures, etc. The development of the material properties may for instance be defined according to Section 5.1.3.

One example where a similar approach have been used is the analyses performed by James and Dollar (2003) for the Portuguese Arch Dam. In their analysis, a transport analysis of the temperature distribution due to hydration was first simulated. In the temperature analysis, the hydration heat was defined as time dependent for each casting segment and the effect of cooling pipes was also considered, as seen in Figure 7-9 . The boundary conditions where successively changed in the model to describe the geometry change due casting, where new elements were introduced/activated for each casting segment with an element "birth" command. In the following mechanical analysis, development of material properties and the extent of cracking was simulated, as illustrated in Figure 7-10.

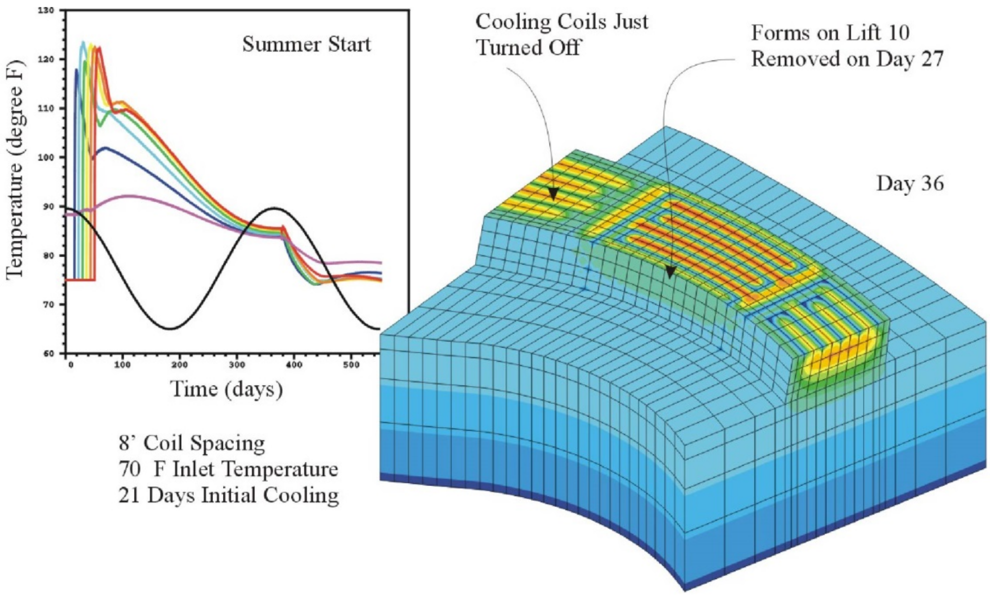


Figure 7-9 Simulated temperatures due to hydration and cooling of the concrete, from James and Dollar (2003).

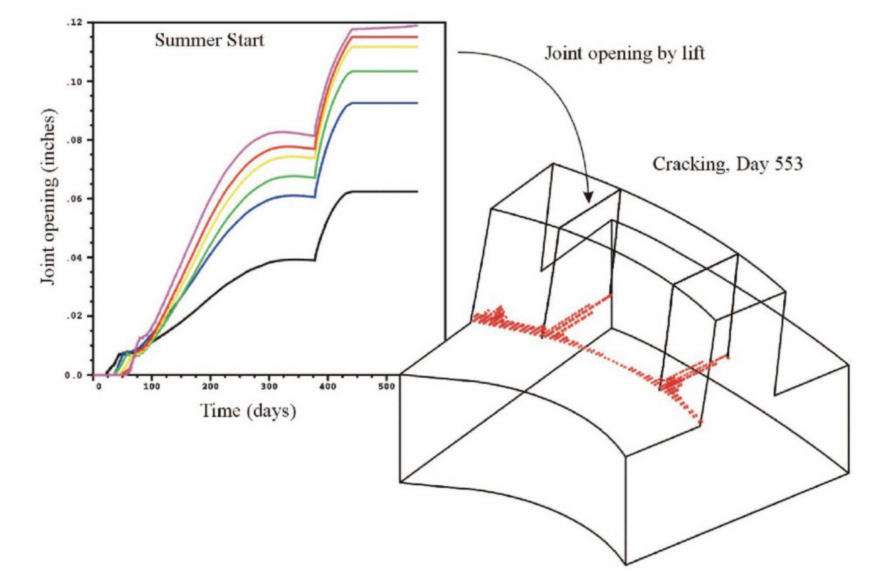


Figure 7-10 Simulated extent of cracking and opening of vertical contraction joints, from James and Dollar (2003).

7.3 ENVIRONMENTAL LOADS

For many large massive concrete structures, environmental effects may the most dominant effects regarding causing cracks.

7.3.1 Temperature

In several cases, it have been shown that seasonal temperature variations may be the reason for cracks found in-situ. This, especially in countries where the temperature may vary significantly over one year, and especially for rather slender structures.

According to design codes, these temperature variations should normally be considered in design where suitable temperature gradients are given depending on the region. For instance, in Eurocode 1-5 (2001), temperature variations are given that should be considered in the design.

According to Cotoi (2015), the seasonal temperature changes of ambient air and the reservoir water propagates only into the first 5 to 6 m from the faces of the dam body, while the daily air temperature variation propagates in a very thin concrete layer (typically maximum of 20 to 30 cm) from the dam face. An example of a temperature distribution through a concrete member is illustrated in Figure 7-11.

Compared with the very large concrete mass in the massive gravity dams, the affected zones are too small to induce significant volumetric changes in the dam, according to Cotoi (2015). Even if some volumetric changes are larger, the contraction joints allows for some free deformation and thereby the horizontal thrust is negligible. However, secondary stresses can occur at the faces of the dam and around openings (drainage galleries, valve chambers) due to temperature differentials. These temperature differentials are caused by differences in the temperature of the concrete surfaces due to ambient air and temperature variation, solar radiation and air movement in openings. These secondary stresses may produce cracks that could lead to progressive deterioration. (Cotoi, 2015)

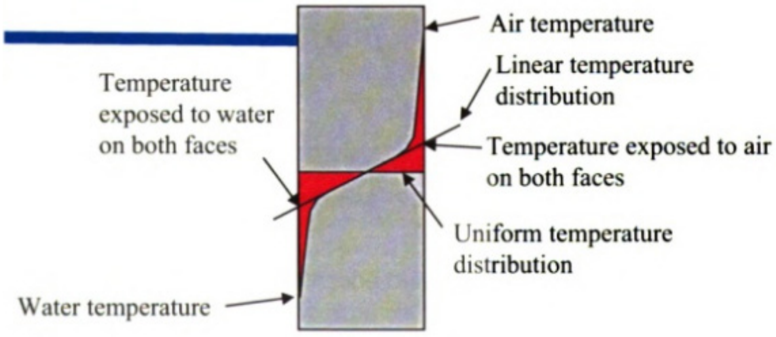


Figure 7-11 Example of temperature distribution in a concrete member, from USBR (2006).

In evaluations of existing dams, for instance to evaluate response of the current status of the dam, the defined maximum and minimum temperatures given in Eurocode 1-5 (2011) could be used. The statistical database from SMHI (http://www.smhi.se/klimatdata [accessed 2016-02-18]) is otherwise a good source to obtain data on historical temperature variations for Swedish dam sites.

For cases where the status of the dam is validated against measurements, it is important that measured temperature data are available in order to capture the actual temperature variations at the site.

Most concrete dams are massive, structures with thick cross-sections, and therefore the convective heat transfer is the most dominant part of the temperature analysis. Cracks may also occur due to radiation from the sun. However, in Sweden the duration that the structure is subjected to warming from solar radiation is rather limited and in most cases not enough to cause more than superficial cracks. Therefore, in many cases radiation may be neglected. However, this has to be determined on a case to case basis.

It is not only the ambient air temperature that is relevant, in addition, temperature variations in the water (dependent on the depth) and if heating of the dam is performed in areas with inspection gangways. Many Swedish buttress dams have insulation walls installed to reduce the thermal gradient over the front-plate and the enclosed area between the front-plate and insulating wall is often heated during the winter months.

Temperatures should as far as possible be defined as convective type of boundary conditions where the ambient air temperatures is defined with a thermal surface resistance resulting in that the temperature is calculated on the surface. If the temperatures instead are defined as prescribed node temperatures, this would result in sharp temperature difference in for instance corners where two temperature boundaries intersect. Node temperature values will also introduce unreasonable large gradients near the surfaces in transient temperature analyses. Thereby, unrealistic stress concentrations will occur in those cases where prescribed node temperatures are used.

When performing analyses of seasonal temperature variations, the daily temperature variations are, as mentioned previously, in general not affecting the structural behaviour. Average temperatures on weekly basis are often sufficient to simulate the structural behaviour. However, this has to be verfied for the specific case. In general, these analyses should be performed as transient and not steady state due to the relatively high thermal inertia of the massive concrete structures. Some examples can for instance be found in Malm (2009), Andersson and Seppälä (2015), Svensen (2016), Léger and Seydou (2009).

An example of a calculated temperature distribution of a concrete buttress dam with an insulating wall is shown in Figure 7-12.

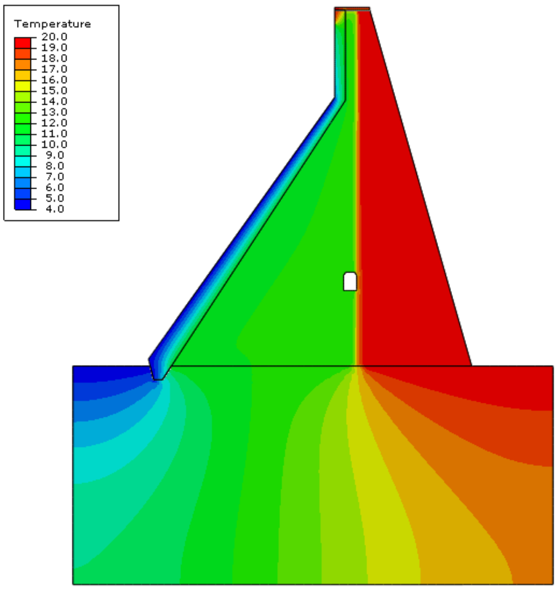


Figure 7-12 Example of a calculated temperature distribution.

For these applications, it is in general sufficient to perform the temperature analysis and the mechanical (stress) analyses as uncoupled. Thereby, the calculated temperature field is introduced as input to the mechanical analysis where the temperature is multiplied with the thermal expansion and results in thermal strain.

$$\varepsilon_{thermal} = \Delta T \cdot \alpha = \left(T - T_{ref}\right) \cdot \alpha$$

where,

$\varepsilon_{thermal}$ is thermal strain [-]

α is the thermal expansion coefficient according to Section 5.1.1 [1 / °C]

T_{ref} is the reference temperature where the concrete is assumed to be stress free according to Section 5.1.1 [°C]

T is the temperature in a specific node [°C]

7.3.2 Creep and Shrinkage

Creep and shrinkage both material related properties and where described in the material chapter in Section 5.1.4 and 5.1.5 respectively.

Based on Eurocode 2

In numerical analyses, shrinkage is often defined as a load effect resulting in reduction in volume. This is usually done where the shrinkage strain is translated into an equivalent temperature, with the use of the thermal coefficient of expansion, i.e. a shrinkage strain equal to 0.3 ‰ corresponds to a temperature of -30 °C. The equivalent temperature due to shrinkage is superimposed with other temperature loads.

However, translating the calculated shrinkage strain according to Section 5.1.4 directly into a temperature is rather conservative. The creep effect will reduce the strains/stresses due to shrinkage. According to Section 5.1.4, the stress can due to a prescribed strain may calculated as

$$\sigma_c = \varepsilon_{cc}(\infty, t_0) \cdot \frac{E_c}{\varphi(\infty, t_0)} = \frac{\varepsilon_{cc}(\infty, t_0)}{\varphi(\infty, t_0)} \cdot E_c$$

where,

$\varepsilon_{cc}(\infty, t_0)$ is the strain at time $t = \infty$ [-]

E_c is the tangent elastic modulus $E_c = 1.05E_{cm}$ [Pa]

$\varphi(\infty, t_0)$ is the creep factor at time $t = \infty$ for the case where the structure was loaded at the concrete age t_0 [-]

Thereby, for a given load resulting in creep deformations, this can be considered in the analysis by either reducing the elastic modulus with the creep factor or by reducing the prescribed strain (for instance from shrinkage) with the creep factor. In FE design analyses, it is complicated to perform analyses where different load effects should be analysed with different stiffness (elastic modulus) of the structure. Therefore, it is conventional to use the expression to the right where each long-term load is reduced with its specific creep factor instead. As an example, Zangeneh Kamali et al. (2013) defines the equivalent temperature difference due to shrinkage as

$$\Delta T_s = \frac{\varepsilon_{cs}}{\alpha \cdot (1 + \varphi)}$$

where,

ΔT_s is the equivalent temperature difference due to shrinkage [°C]

ε_{cs} is the total shrinkage strain according to Section 5.1.4 [-]

α is the coefficient of thermal expansion according to Section 5.1.1 [°C^{-1}]

φ is the creep factor according to Section 5.1.5 [-]

The problem is however, to define the time when the creep starts to since the shrinkage is an ongoing process. If the creep factor is determined for when the load starts, this results in an overestimation of the creep effect and thereby underestimates the effect of shrinkage. As the example shown previously in Figure 5-11, the creep factor for loading after one year could be about half the value of the creep factor for loading after 7 days.

If this approach is used, it is reasonable to treat the autogenous shrinkage and the drying shrinkage separately with two different creep factors, since the autogenous shrinkage occurs in general for only the first year/years while the drying shrinkage occurs over a very long time.

Another approach to simulate the creep, is that the additional strain due to creep is calculated according to Section 5.1.5 for each long-term load. This strain contribution can then be added together for all loads. This approach may however, be straightforward when analysing simple structures and load with almost constant stress/strain state but more difficult when using more complex cases.

If more detailed analyses of creep behaviour is required, creep should instead be modelled with a viscoelastic material model. A simple example of how this could be performed is presented Malm and Sundquist (2010). More detailed methods are available in the literature to include the different aspects of concrete creep, such as the Microprestress-Solidification theory by Bazant et al. (1997).

Simulation of drying of concrete

As mentioned in Section 5.1.4, the shrinkage can also be simulated in more detailed manner where a moisture transport analysis is performed to simulate the drying of concrete.

In most general multi-physics FE-software's it is possible to perform both transport analyses and mechanical analyses. It is often possible to define how these analyses should be coupled, and thereby how the parameters in the two different analyses are dependent on each other.

In the simplest form of coupling, one-way coupling, one analysis is performed first and the quantities obtained from this is introduced as input in the second analysis. One example of this one-way coupling that is commonly used is the one-way coupling where the temperatures obtained from a transport analysis influence the stresses in a mechanical analysis, but not vice versa.

In analyses with mutual coupling, the quantities are obtained from the two analyses are co-dependent on each other where the analysis has to be performed sequential or parallel as a co-simulations. One example of a case where this might be necessary to consider is for instance that the moisture transport is dependent on the temperature conditions and vice versa.

When calculating moisture transport in concrete, the moisture content or the relative humidity are often defined as the driving potential. It is also possible to use other alternatives such as pore pressure, water saturation etc. The choice of driving potential depends on how the moisture transport is considered in the analysis. The most common case is that the moisture transport is considered to occur through diffusion, which is govern by Fick's law. An alternative approach is to consider the moisture transport to occur through capillary transport, which is govern by the pore pressure gradient and modelled with Darcy's law. In some cases, it may also be needed to combine both of these approaches. This may for instance the case if the moisture transport occurs for a case subjected a temperature gradient.

As mentioned in Section 5.1.4, one simple approach to calculate the moisture transport in concrete due to diffusion of water vapour based on fib Model Code (2010).

After the moisture content has been calculated, the shrinkage strain can be calculated for instance as described in Section 5.1.4. In the project by Malm et al (2013a) a simple approach was used to simulate the relative humidity in concrete due to drying and based on this the shrinkage was calculated, as shown in Figure 7-13. In this project, analyses where performed to explain possible reasons for the cracks found in-situ of a concrete structure which acts as the foundation for the generator in a hydropower plant.

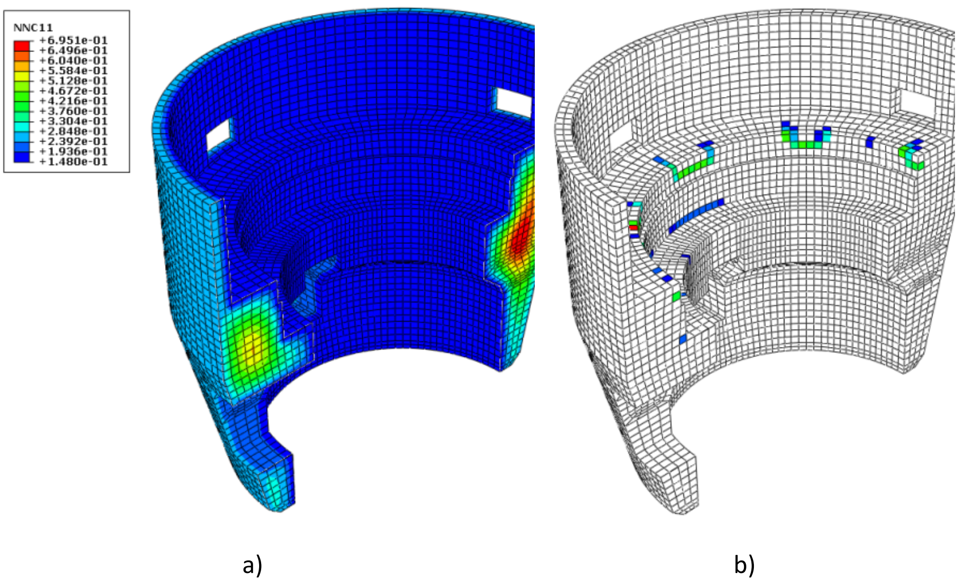


Figure 7-13 Calculation of distribution of relative humidity (a) after 20 years and the extent of cracking due to drying shrinkage (b). From Malm et al. (2013b)

In more detailed analyses with fully coupled hygro-mechanical models, it is possible to also include how the cracking affects the permeability of the concrete which was performed in the project by Gasch et al (2016) as illustrated in Figure 7-14. In this project, the drying shrinkage of a concrete gravity dam was analysed as it was subjected to cyclic seasonal temperature variations.

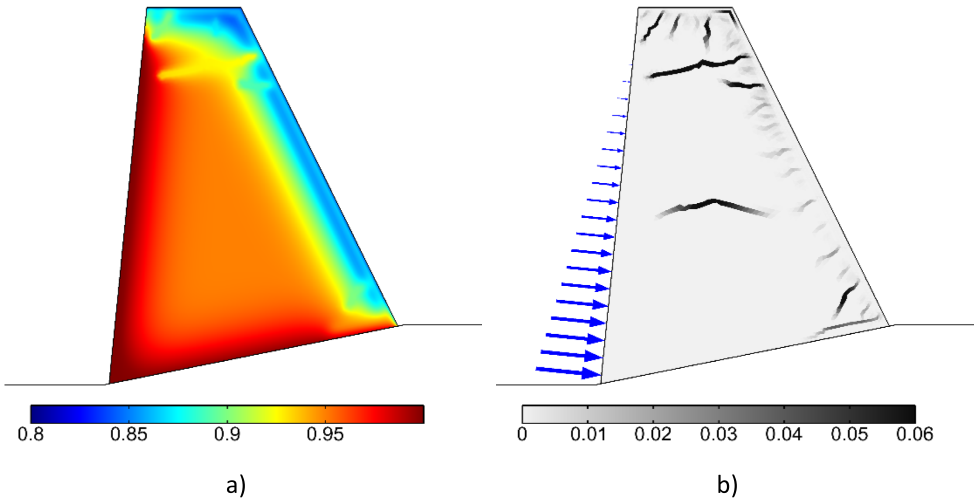


Figure 7-14 Calculation of relative humidity (a) and extent of cracking (b) of a concrete gravity dam subjected to drying and cyclic seasonal temperature variations. From Gasch et al (2016).

8 Seismic analyses

Seismic events are in most countries the dominant design loads. There are therefore many good guidelines regarding seismic design and FE-analyses of concrete dams, as previously presented in Section 2.5.

In Sweden, seismic loads do not have to be considered in design of concrete dams according to RIDAS (2011), since it is considered as a low seismic area. Therefore, this section will be very brief regarding seismic analyses and if deeper knowledge is needed then it is suggested to read some of the references presented previously, such as ; USACE (1999), USACE (2003) and USACE (2007).

8.1 SEISMIC SIGNALS

Measurements of seismic ground motions are performed with accelerometers and results in accelerograms, i.e. time history signals for the variation in acceleration. The ground motion of a site depends on its geology, the distant from the fault, the earthquake magnitude, source characteristics, the geology along the travel paths of the seismic wave etc. (Rydell 2014)

Based on recorded time histories, design response spectras have been determined for different regions and these are in general given in design codes. Design response spectras do not correspond to one specific earthquake but instead as an envelope of many possible earthquakes at the region. An example of a response spectrum is shown in Figure 8-1. This response spectra corresponds to the horizontal ground motion according to the 1e-5 earthquake for southern Sweden according to SKI (1992).

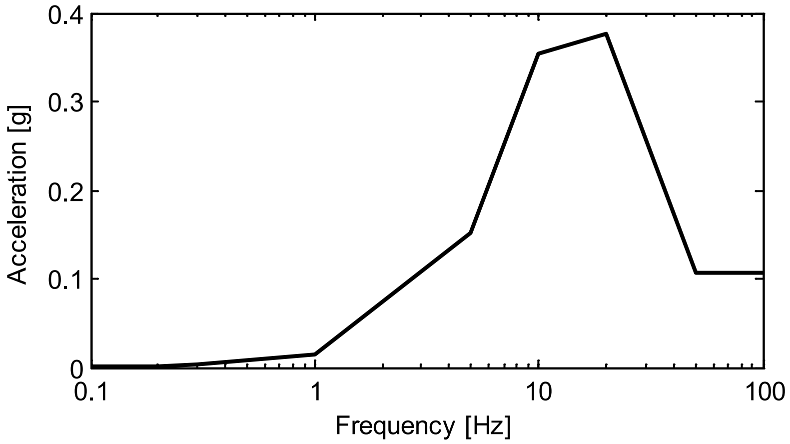


Figure 8-1 Example of a response spectrum.

In Eurocode 8 (2009) response spectras are given for different soil classes. The return period of these, earthquakes are 475 years. However, for concrete dams, much higher return periods are considered in design. According to ICOLD (2010), the revised version of Bulletin 72, the following definition of earthquakes exists

- The Safety Evaluation Earthquake (SEE) – is the maximum level of ground motion for which the dam should be designed or analysed, i.e. it should not lead to uncontrollable release of water. This corresponds to a very long return period, for example 10 000 years.
- The Operation Basic Earthquake (OBE) – is the level of ground motion at the dam for which only minor damage is acceptable. The dam, appurtenant structures and equipment should remain functional. In many cases, OBE earthquakes are chosen with a minimum return period of 145 years (i.e. corresponds to 50 % probability of not being exceeded in 100 years)
- The Reservoir-Triggered Earthquake (RTE) – represent the maximum level of ground motion that can be trigged at the site due to filling, drawdown or the presence of the reservoir.

Another term that is commonly used in the literature is the Maximum Credible Earthquake (MCE), and it is the largest conceivable earthquake magnitude that is considered possible.

Recorded earthquake signals does not cover the whole range of possible conditions, and therefore synthetic or artificial ground motions representing any earthquake size and the seismicity of a certain site must often supplement these records, according to ICOLD (2010). Several techniques to generate synthetic accelerograms based on response spectras exists. Time histories have three important characteristics; amplitude, frequency content and duration. An example of a synthetic time history is shown in Figure 8-2. This acceleration time history is derived from the response spectra shown in Figure 8-1.

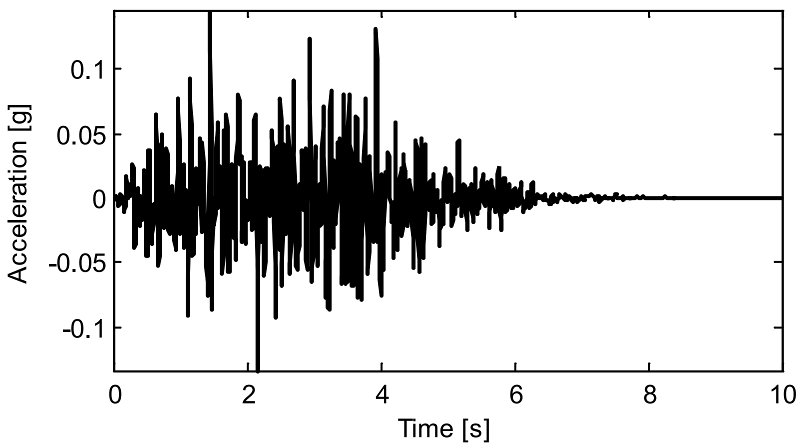


Figure 8-2 Example of a synthetic acceleration time history signal.

8.2 DYNAMIC ANALYSIS METHODS

Seismic analyses may in general be performed with methods based on either;

- Mode superposition
  - × Response spectrum analysis
  - × Modal dynamic time history analysis
- Direct time integration.

In this section, a brief description of the difference between these two techniques is presented.

8.2.1 Mode superposition metods

Mode superposition methods may be used if the system is classically damped (i.e. diagonal damping matrix) and that the structural response is linear. This means that nonlinear effects such as; material nonlinearity, interactions/joints etc., cannot be considered. The structural response in this method is calculated as the summation (superposed) of the response from each mode. In this report, two different approaches for mode superposition methods are presented;

- Response spectrum analysis
- Modal dynamic time history analysis.

As a first step in both methods, a calculation has to be performed to extract the natural frequencies and eigenmodes. The response of the dam can then be calculated with either of these methods based on the extracted modes and mode shapes, which describe the shape of the displacements for each mode.

In the response spectrum analysis, the seismic action is described through a design response spectrum. In this approach, only the peak response, i.e. maximum deflections, stresses etc. are obtained. There are several different approaches to combine the quantities from the different modes, and the FE-analysist have to make sure that a suitable mode combination method is used. The response spectrum analysis is often used in design since it is a quick method. However, it may be difficult to evaluate maximum and minimum principal stresses when a response spectrum analysis is used since the combination of results from the different modes in general is based on the root of squared quantities (in procedures such as SRSS[4] or CQC[5]) and hence the sign (i.e. indicating tension or compression) is lost, Aswegan and Charney (2014).

In modal dynamic time history analysis, a time history signal is used to describe the seismic load instead of a response spectra. In this approach, results will be obtained for all time increments, typically with a predefined time step that is dependent of the highest frequency of interest. The size of the predefined time step corresponds to the Nyquist frequency as follows

$$f_{max} = \frac{1}{2\Delta t}$$

where,

f_{max} is the Nyquist frequency, i.e. maximum frequency that can be extracted [Hz]

Δt is the predefined time step used in the analysis [s]

[4] Square Root of the Sum of the Squares
[5] Complete Quadratic Combination

One important aspect of using either of the mode superposition methods is to make sure that sufficient modes are considered in the analysis. In order to determine suitable number of modes that should be considered, the cumulative effective mass should be studied. Each mode is associated with an amount of effective mass that is excited in each direction. The cumulative sum of the effective mass for all considered modes should be high enough to make sure that all important modes are included. The cumulative effective mass should high enough to cover all modes of interest. As an example, this could for instance be higher than 90 % of the total mass in all directions.

8.2.2 Direct time integration methods

In direct time integration methods, the equation for dynamic equilibrium is solved directly. This approach can be used when the system has nonlinearities (e.g. material, interaction or geometrical nonlinearities) that needs to be considered. This method uses a numerical time-stepping procedure to solve the coupled system of differential equations of motion. The results will be obtained for all time increments (as in the modal dynamic analysis), typically with a predefined time step (sampling frequency) that corresponds to the Nyquist frequency, i.e. it is equal to twice the highest frequency that can be extracted.

The guideline developed by USACE (2003) is devoted to time history analyses of hydraulic concrete structures.

8.3 MASSLESS ROCK APPROACH

In general, arch dams, gravity dams and sometimes lock walls and intake towers are built on competent rock foundation and in these cases it is considered sufficient to describe the rock as massless in the finite element analyses. The size of the rock mass should be of comparable size of the structure as discussed previously in Section 6.1.

The earthquake input is applied directly at the fixed boundaries of the massless foundation model (previously shown in Figure 6-3), as illustrated in in Figure 8-3.

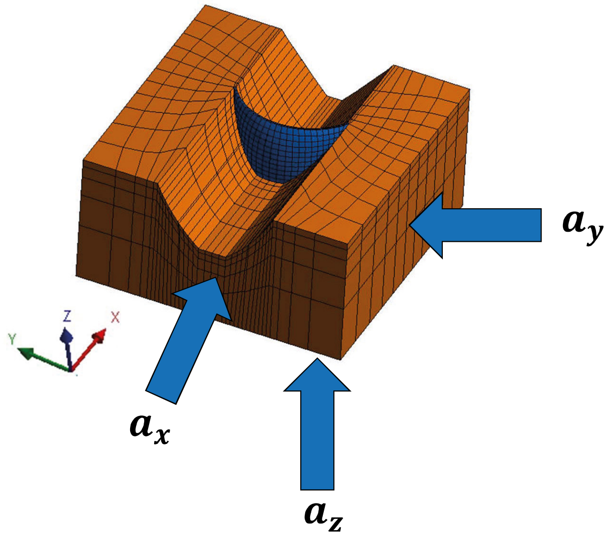


Figure 8-3 Seismic ground motions are applied at the outer surfaces of the rock mass, from Shahriari (2013).

8.4 DYNAMIC PROPERTIES

8.4.1 Dynamically increased material properties

In seismic analyses, it is common in many design codes that the elastic modulus and the strength of the concrete should be increased, typically 25 %.

For instance, in the Swiss directives for seismic analyses (Swiss Committee on dams, 2003), the elastic modulus may be increased with 25 % while the compressive and tensile uniaxial strength may be increased with 50 %.

8.4.2 Damping

Damping of a structure is the result of nonlinearities that cause dissipation of energy due to friction in cracks, etc. The choice of damping has a significant impact on the results from dynamic analyses. Suitable values for critical damping factor are specified in different design codes and depends on the type of material, intensity of the earthquake etc, but is often between 5 and 10 % for concrete.

In the modal superposition methods, it is possible to define modal damping, i.e. all frequencies are defined with the same damping factor.

This is not possible in analyses based on direct time integration methods. In direct time integration methods, Rayleigh damping is used. Rayleigh damping is defined by two factors α and β which corresponds to material and stiffness proportional damping coefficients, as seen in Figure 8-4. The principle is that two frequencies are determined (ω_n *and* ω_m) for which damping factors are specified (ξ_n *and* ξ_m) and the damping for the whole frequency band is then calculated. The determination of these two natural frequencies is normally based on experience, the effective mass under consideration and compromises in the damped frequency ranges, Goldgruber (2015).

It should be noted, that the damping is significantly overestimated for frequencies lower than the lower bound and higher to the upper bound. For the frequencies within the interval of interest, the damping is more or less underestimated.

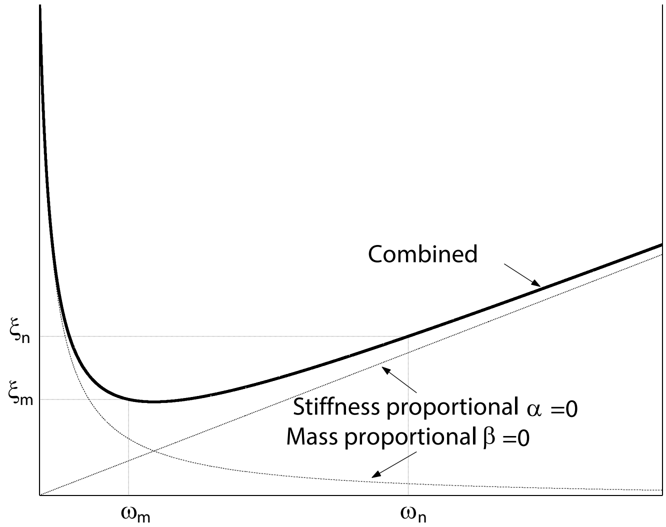


Figure 8-4 Rayleigh damping, reproduced from Clough and Penzien (1993).

The conventional way that Rayleigh damping is used in most references in the literature, is that for instance the first and third (first and fifth or first and seventh) natural frequencies are chosen as the two bounds. This approach is however not to be recommended since it may significantly overestimate frequencies of interest, since the eigenmodes are usually rather close to each other.

Hellgren (2014), studied different approaches to determine the Rayleigh damping, where three approaches were used to determine the two bounds for the frequencies.

1. The first (1.5 Hz) and third eigenmode (2.1 Hz)
2. Based on frequencies corresponding to 5 and 90 % of the cumulative mass (1.0 and 10.0 Hz respectively)
3. Based on the method developed by Spears and Jensen (2012) where the Rayleigh damping is iterated to obtain results that correspond to the response from constant modal damping. The iterated frequencies were (2.1 Hz and 6.6 Hz)

The calculated Rayleigh damping curves are presented in Figure 8-5 for the three approaches and in addition a case with 5 % constant damping for a frequency up to 50 Hz. As it can be seen in Figure 8-5, the damping varies significantly over a frequency interval between 0 and 50 Hz, where a damping factor of 70 % is used in the first approach for frequencies of 50 Hz. In the third, conservative, approach the corresponding damping at 50 Hz is about 20 %. It can also be seen that the conservative approach (no 3), it underestimates the damping in the interval between the two selected frequencies i.e. $1.0 \text{ Hz} < f < 10 \text{ Hz}$.

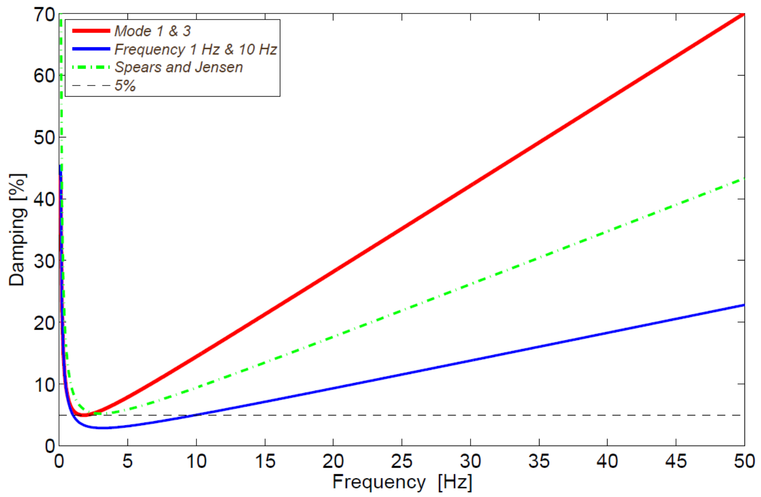


Figure 8-5 Rayleigh damping depending on the choice of the two frequencies. From Hellgren (2014).

The calculated response of the crest is presented in Figure 8-6 for the three approaches and for an analysis performed with constant modal damping of 5 %. As seen in the figure, the traditional approach of selecting the first and third mode significantly underestimates the response for frequencies above about 5 Hz, while the conservative approach significantly overestimates the response above about 5 Hz. The approach based on Spears and Jensen (2012) corresponds well to the analysis based on modal damping.

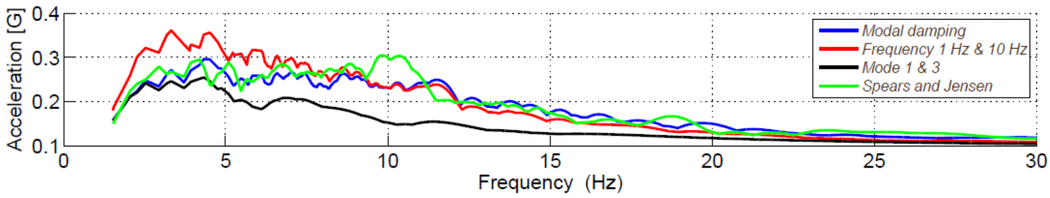


Figure 8-6 Response spectra of the crest acceleration depending on the choice of the two frequencies in Rayleigh damping. From Hellgren (2014).

Thereby, the choice of critical damping factor but also on how it is translated into Rayleigh damping has a significant effect on the results.

8.5 FLUID-STRUCTURE-INTERACTION

There are many, more or less advanced, approaches to model fluid-structure-interaction (FSI), see for instance Gasch et al (2013) for further reading. In the case with FSI between a concrete dams and the reservoir, it is mainly two techniques that are used

- Added mass - In the added mass technique, usually the principle developed by Westergaard (1933) is used. This approach is however, based on the assumptions that the dam is rigid, the water can be considered as an incompressive fluid and it is only developed for a vertical upstream face.

Further developments of this approach for non-vertical upstream faces, with the Generalized Westergaard added mass (developed by Kuo, 1982) is for instance described in Goldgruber (2015).

- Acoustic elements - Acoustic fluids or acoustic elements in finite element analyses are special purpose elements which are describing the pressure distribution over time in an acoustic media like water. the fluid is compressible (density changes due to pressure variations)

In most cases, it can be considered sufficient to use the Generalized Westergaard added mass approach.

The influence of incompressible and compressible reservoir was studied by Chopra (1968) where it was found that the key parameter governing if the compressibility of the reservoir has to be taken into account is the ratio between the natural frequency of the reservoir and the natural frequency of the dam. The frequency of the reservoir may be calculated as follows

$$f_r = \frac{c_w}{4h}$$

where,

f_r is the natural frequency of the reservoir [Hz]

c_w is the wave propagation velocity [m/s]

h is the water depth [m]

According to Dabre (2000), the wave propagation velocity of water is equal to 1451 m/s. Thereby, as an example it can be mentioned that the natural frequency of the reservoir is about 9 Hz for a dam with $h = 40$ m and about 2.4 Hz for a dam with $h = 150$ m.

The ratio of the natural frequencies are defined as follows

$$\Omega_r = \frac{f_r}{f_s} \begin{cases} < 2.0 \ \text{compressible reservoir} \\ > 2.0 \ \text{incompressible reservoir} \end{cases}$$

where,

f_s is the natural frequency of the dam [Hz]

The consideration of a compressible reservoir for modelling dam-reservoir interaction is recommended for most problems, because the response might be significantly underestimated otherwise. (Goldgruber, 2015)

Despite this, it is very common that the added mass approach is used in design and evaluation projects. The added mass approach is in general considered to be conservative.

8.6 GENERAL ASSUMPTIONS

The following general assumptions are often used in seismic analyses since they are in general considered sufficient accurate and conservative

- The general Westergaard approach are often considered sufficient to describe the hydrodynamic forces.
- The length of the reservoir (if modelled) should be at least twice as long as the dam height in order to be considered as infinite. Some publications even shows that the reservoir must be up to 3 or 4 times the dam height in order to be considered as infinite, Chopra (2008)
- In many cases, the worst seismic case for arch dams is often for cases with a half-full reservoir rather than a full.
- The rock is in general defined as massless, i.e. with zero density. The reason for this is to prevent wave reflection on the boundaries.
- Considering bottom absorption may have a significant effect on the results as presented by Hellgren (2014). In addition, large uncertainties are often present regarding the size of reflection coefficient.
- Seismic excitation is defined as prescribed accelerations on the outer rock surfaces, only the direction perpendicular to the surface is exited, as seen in Figure 8-3.

9 Solution techniques

Solving nonlinear analyses are far more computational expensive than solving the corresponding linear analyses. This is because it is not possible to solve the equation directly in a nonlinear analysis, since the relation between force and stiffness depends on the degree of nonlinearities considered. Instead, the equation has to be solved in an iterative manner, where the prescribed load or displacement is divided into increments and is successively increased up to the desired load. The structural response is determined iteratively for each increment. An example of a load and deflection curve for an analysis with nonlinear material properties is illustrated in Figure 9-1.

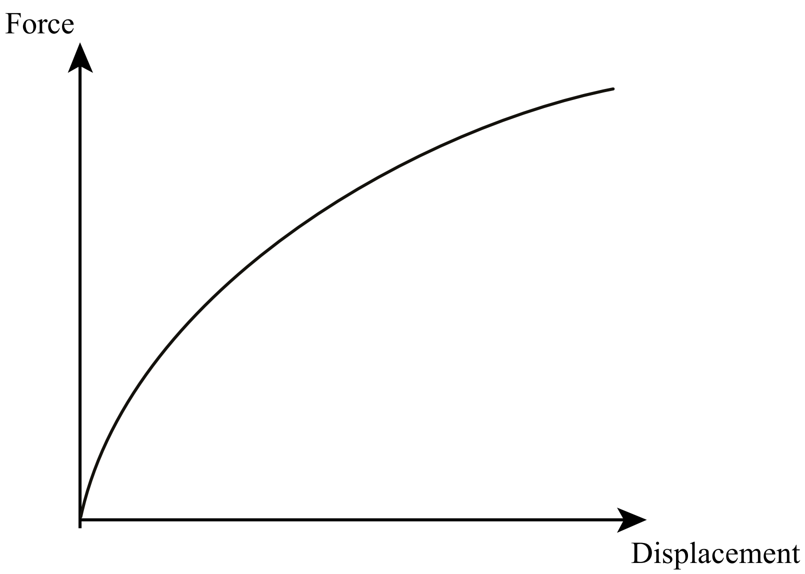


Figure 9-1 Nonlinear load and deflection curve.

The intention of this section is to give a brief overview of common solution techniques that are used in the software's to solve nonlinear behaviour. Other iterative solution techniques also exists, but are out of the scope of this report. The main aspect when dealing with concrete dams is material nonlinearities, but other sources of nonlinearities may be caused by large deformations, stability or geometric nonlinearities.

The types of analysis that can be used are often categorized as load-controlled, displacement-controlled or quasi-static. However, regardless of solution technique all are based on solve the system

$$\mathbf{F} = \mathbf{K}_t \cdot \mathbf{u}$$

where,

$\mathbf{K}_t$**t** is the tangent stiffness matrix

$\mathbf{F}$ is the force vector

$\mathbf{u}$ is the nodal displacement vector.

In a dynamic system, the equation also includes the inertia forces ($\mathbf{M} \cdot \ddot{\mathbf{u}}$) and the viscous forces ($\mathbf{C} \cdot \dot{\mathbf{u}}$)

$$\mathbf{F} = \mathbf{M} \cdot \ddot{\mathbf{u}} + \mathbf{C} \cdot \dot{\mathbf{u}} + \mathbf{K}_t \cdot \mathbf{u}$$

where,

$\mathbf{M}$ is the mass matrix

$\mathbf{C}$ is the damping matrix

$\ddot{\mathbf{u}}$ is the nodal acceleration vector

$\dot{\mathbf{u}}$ is the nodal velocity vector

The amount of nonlinearity that the dam is subjected to is described by the tangent stiffness matrix and the system has to be solved with an iterative technique since the tangent stiffness depends on current and previous load levels.

9.1.1 Newton Raphson method

The most commonly used technique to solve nonlinear systems of equations is the Newton Raphson method.

The principle used is based on successive iteration of the tangent stiffness of a given load-displacement function to find the corresponding displacement. The Newton Raphson method is, thereby, based on a load-controlled method, where the load increment is successively increased up to the desired load level and the corresponding displacement is iterated for each load level.

For a given nonlinear function, $f(u)$, the displacement can be iterated according to the following expression

$$u_{i+1} = u_i - \frac{f(u)}{\frac{\mathrm{d}f(u_i)}{\mathrm{d}u}}$$

With this approach, each iteration results in increased accuracy for the estimation of the displacement. This approach is applied to solve the system $\mathbf{F} = \mathbf{K}_t \cdot \mathbf{u}$ with respect of $\mathbf{K}_t$ by increasing the load $\mathbf{F}$. In Figure 9-2 the iteration procedure when using the Newton Raphson technique is illustrated. As illustrated in the figure, the load is about to be increased from a previous load value, here denoted as F_0, to the new load level, here denoted as F_1. In order to do this, the first iteration is performed with the tangent stiffness matrix as K_0 (based on the initial state as $u_0 = 0$), resulting in an estimated displacement u_a at point a'. In this case, the external force F_1 is not equal to the internal forces I_a (at point a) resulting in an error $e_a = F_1 - I_a$.

In order for the structure to be in equilibrium, the sum of all forces (and moments) in each node has to be zero. The structure is thereby in equilibrium if the error is equal to zero. This is, however, seldom the case in nonlinear analyses and therefore tolerances have to be defined regarding an accepted limit of the size of the error. If the error is larger than the predefined tolerance, a new iteration has to be performed with an updated value of the stiffness. This is done by reforming the tangent stiffness matrix at point a. With the updated tangent stiffness matrix a new estimation is performed for the load increment F_1. Based on this a new estimated displacement u_b at point b' is obtained. Once again, the external force F_1 is compared to the internal forces I_b (at point b) which results in the error $e_b = F_1 - I_b$. If the error is lower than the predefined tolerance, the solution is considered acceptable and the analysis can be performed for the next load increment level F_2.

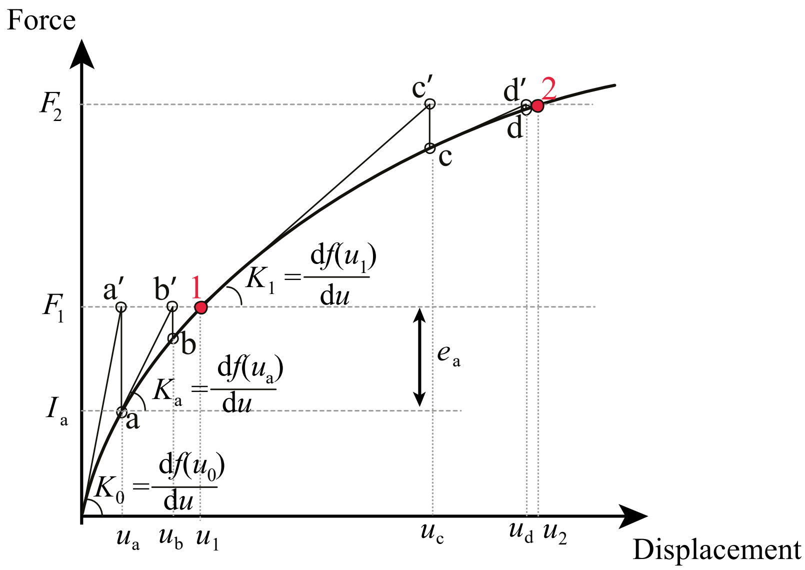


Figure 9-2 Illustration of iterations for two increments in a nonlinear analysis with the Newton Raphson's method. Reproduced from Cook et al (2002).

The principle with updating and factorising the tangent stiffness matrix prior to each iteration is time-consuming and therefore there are alternative approaches where the tangent stiffness matrix is updated more seldom. An example of this the Modified Newton Raphson technique where the tangent stiffness matrix is only updated in the beginning of each increment and maintained for all subsequent iterations within this increment. This is illustrated in Figure 9-3. This approach requires more iterations than the Newton Raphson methods, since the estimation of the tangent stiffness matrix is not redefined for each iteration. With this method, the total cpu-time required may decrease even though the number of iterations increases as long as the solution is not significantly nonlinear.

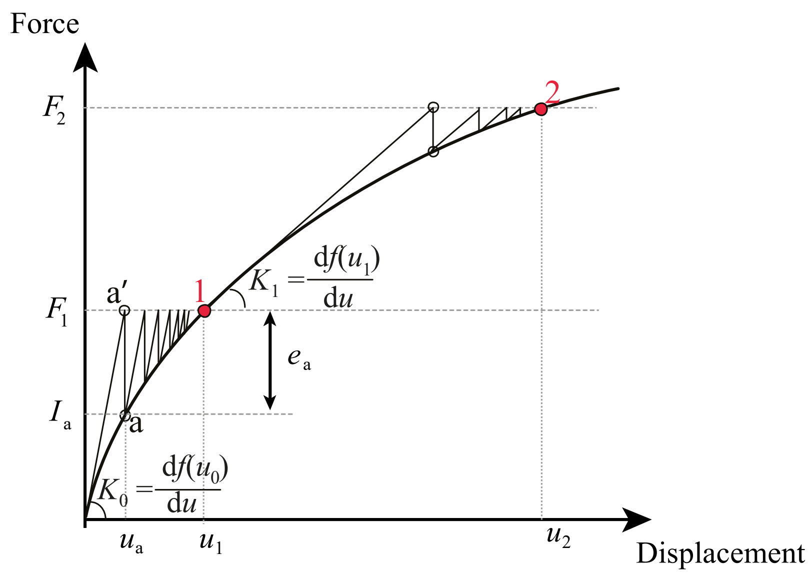


Figure 9-3 Illustration of iterations for two increments in a nonlinear analysis with the Modified Newton Raphsons method. Reproduced from Cook et al (2002).

In order to improve convergence and improve the solution stability, it is also possible to use a line search algorithm. When using a line search algorithm, the definition of the stiffness matrix is based on a fraction of the previous solution, i.e. where the stiffness

$$\mathbf{K}_{i+1} = \mathbf{K}_i + \mu\Delta\mathbf{K}$$

where,

μ is a constant obtained from the line search algorithm, $0 < \mu < 1$, and applied to the increment $\Delta\mathbf{K}$.

9.1.2 Arc-length method

The Newton Raphson iteration technique is based on successively increasing loads. These methods may have problems to predict the post-peak behaviour if the structure is subjected to a softening behaviour. If the loading is performed with successively increasing load increments, the iteration will fail directly as softening occurs. However, in many cases, it might be possible to capture softening behaviour with this technique if the load consists of a prescribed displacement, since increasing displacements may result in reduced loads.

Another approach that may be used to iteratively calculate the softening response of a structure is the arc-length method. The arc-length method is a form of Newton Raphson iteration, but combines both load and displacement incrementation. The path of the iteration follows a circle with a length Δl as illustrated in Figure 9-4. In the figure, three iterations (a, b and c) are shown. The stiffness of each iteration is based on the stiffness from the previous iteration (i.e. the inclination of the curve). Since the iteration is performed along a circular path, it can capture a descending load and deformation curve and. This approach can also be used to capture a snap-through behaviour, i.e. where both force and displacements have to be decreased from one iteration to the next in order to maintain equilibrium.

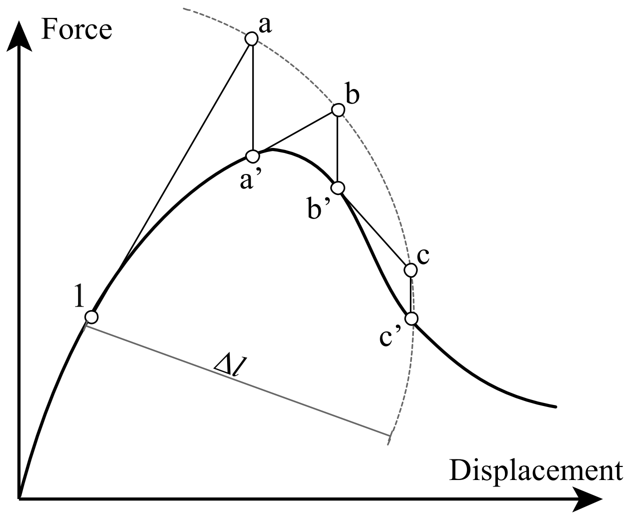


Figure 9-4 Illustration of the arc-length method, reproduced from Cook (2003).

9.1.3 Quasi-static analyses

Performing nonlinear analyses where the material nonlinearity of concrete is considered often results in problems with finding a converging solution and thereby the iterations fails prematurely. This may happen even if a displacement controlled iteration based on full Newton-Raphson or the Arc-length iteration procedure is used.

In these cases, it can be beneficial to perform the analysis as quasi-static instead of an ordinary static analysis. Quasi-static analyses refers to a technique where a dynamic solver is used to solve a static numerical problem where the load is applied slow enough to minimize the kinetic energy and thereby reducing significant inertia effects. For problems involving brittle failure, this is especially important since the sudden drop in load carrying capacity that accompanies a brittle failure generally leads to increased kinetic energy content of the response.

Quasi-static analyses may be performed with either an implicit (see Section 9.2.1) or an explicit (see Section 9.2.2) solver. Since these dynamic solvers are used, inertia effects are thereby allowed and these help regularize unstable behaviour in the model such as temporarily unconstrained rigid body modes or "snap-through" phenomena. Considerable numerical dissipation may be required to obtain convergence during certain stages of the loading history, (Dassault 2014).

In dynamic solvers, the time of the analysis step has a physical meaning. In order to keep the quasi-static analyses computational effective this time step should be as short as possible as long as the kinetic energy can be kept at a reasonable low level throughout the analysis. Another approach that can be used in explicit analyses is to use mass scaling.

The stable time increment in explicit analyses is proportional to the square root of the material density. Thereby, by increasing the density with a factor f^2 then the time increment is increased with a factor f. Mass scaling can be used where the density is increased directly but then gravity loads have to be adjusted accordingly if these are included in the analysis. Some software's allows for definition of mass scaling that is only used for the solver and does not affect the gravity loads.

Because the time of the analysis step has a physical meaning in quasi-static analyses, then it is not recommended to increase displacements or load linearly. If it is possible to model the loads as prescribed displacements, then one method is to use prescribed velocity instead to ensure low kinetic energy initially in the analysis. Otherwise, an amplitude function which uses a slow loading rate initially and near the failure should be used. This amplitude function could for instance be as shown in Figure 9-5. This smooth amplitude function can be calculated as follows according to Dassault (2014)

$$A = A_0 + (A_1 - A_0) \cdot \xi^3 \cdot (10 - 15 \cdot \xi + 6 \cdot \xi^2)$$

with

$$\xi = \frac{t - t_0}{t_1 - t}$$

where,

A is the amplitude and A_0 and A_1 are the desired amplitude values at the beginning and end of the analysis step respectively

t is the time function and t_0 and t_1 are the time values at the beginning and end of the analysis step respectively

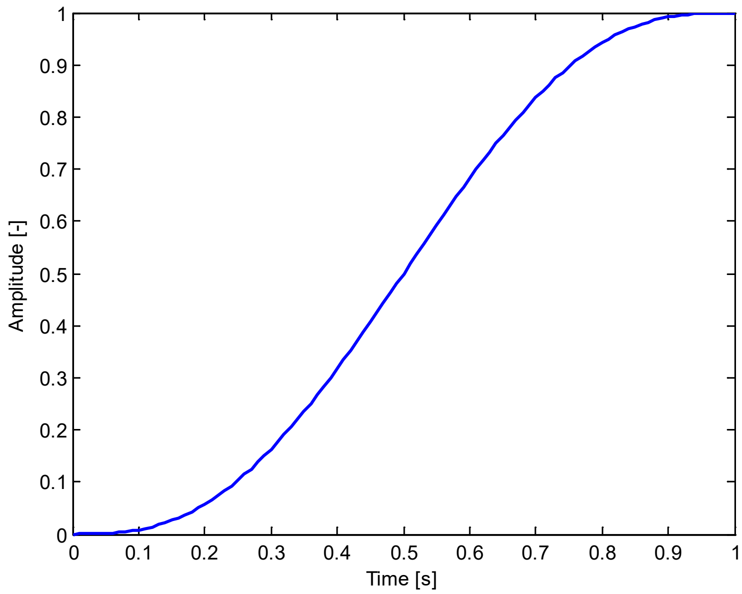


Figure 9-5 A smooth amplitude function used for quasi-static analyses.

9.2 NONLINEAR DYNAMIC ANALYSIS

Iterative solution methods for dynamic problems are in general based on Newmark Beta Methods. A general step method was proposed by Newmark and the equations for the velocity and displacement in an increment i is defined as

$$\dot{u}_i = \dot{u}_{i-1} + (1-\gamma) \cdot \Delta t \cdot \ddot{u}_{i-1} + \gamma \cdot \Delta t \cdot \ddot{u}_i$$

and

$$u_i = u_{i-1} + \Delta t \cdot \dot{u}_{i-1} + \left(\frac{1}{2} - \beta\right) \cdot \Delta t^2 \cdot \ddot{u}_{i-1} + \beta \cdot \Delta t^2 \cdot \ddot{u}_i$$

where,

Δt is the time increment.

γ is a constant that provides a linear weight factor between the importance of the initial and final acceleration on the velocity.

β is a constant that provides a linear weight factor between the importance of the initial and final accelerations for the displacement.

According to Clough and Penzien (1993), the constant γ governs the amount of artificial damping that is introduced into the analysis and for the case $\gamma = 1/2$ no artificial damping is introduced. Depending on how, these constants are chosen, implicit or explicit integration scheme is obtained, as illustrated in Table 9-1.

Table 9-1 Integration constants for different Newmark methods.

Method	Type	γ	β
Trapezoidal integration	Implicit	1/2	1/4
Linear acceleration	Implicit	1/2	1/6
Central difference	Explicit	1/2	0

The main difference between implicit and explicit integration, is that in the explicit integration is that the solution of one increment is not dependent on the obtained accelerations from the previous increment (since $\beta = 0$).

The stability criteria for the size of the increment in Newmark methods is defined as

$$\frac{\Delta t}{T_n} \leq \frac{1}{\pi\sqrt{2}} \cdot \frac{1}{\sqrt{\gamma - 2\beta}}$$

where,

T_n is the natural period (i.e. the inverse of the natural frequency) [s].

Using the implicit approach based on the trapezoidal integration rule with $\gamma = 1/2$ and $\beta = 1/4$ results in an approach that is unconditionally stable, i.e. stable for all time increments

$$\frac{\Delta t}{T_n} \leq \infty$$

9.2.1 Implicit dynamics algorithm

The implicit solution scheme is an unconditionally stable approach where calculation of quantities in one increment is based on the quantities calculated in the previous increment. This allows the solver to use relatively large increments. However, each increment is computational expensive where the tangent stiffness matrix has to be formed and inverted in order for the equations to be solved.

9.2.2 Explicit dynamics algorithm

Explicit analyses, are on the other hand conditionally stable and hence requires that very small increments (which are dictated by a stability limit) are used. The use of small increments is advantageous since it allows the solution to proceed without iterations and without requiring tangent stiffness matrices to be formed and in addition, it simplifies the treatment of contact conditions.

Since there is no need to define a tangent stiffness matrix and invert it, the explicit solution technique is computationally effective for large models.

10 Evaluation of results

The results from all numerical analyses must be verified and validated. Numerical analyses performed with the finite element method or finite difference method are always approximations of a behaviour and is highly dependent on the choices and assumptions made by the FE-analysist.

In this section, recommendations are given on types of validations and verifications that should be performed in projects with numerical analyses. In addition to this, some suggestions on what to look for when evaluating results are given in the end of this section.

10.1 VALIDATION OF RESULTS

All numerical models should be validated. The complexity of the model, or if the FE-analysist is applying a feature that has not been used previously determines the level of validation that is required. The recommendations given in this section corresponds to the recommendations given in the report by Ekström et al. (2016) intended for review of numerical models of hydraulic structures.

- Even simple models based on conventional well-proven procedures, needs to be validated. In these cases, it is primarily the choices made with regard of the method/model that should be validated. This could be tested with sensitivity analyses, where the results from different choices are compared in order to study the influence of these choices.
- In cases, where user-defined material models or new (for the FE-engineer) material models are used or complex models based on advanced features, then extensive validation by experiments and/or benchmarks should be performed in addition to the tests mentioned in the bullet above.

According to fib Model Code (2010), the following types of validation should be performed;

- Basic material test (see Section 10.1.1)
- Structural test (see Section 10.1.2)
- Mesh sensitivity tests (see Section 10.1.3)

In addition to these bullets, one other important type of validation is always required - verification analyses. Verification analyses are further described in Section 10.1.4.

10.1.1 Basic material tests

The purpose of the basic material tests is to validate the constitutive relations and should be performed on simple structural examples. The aim of using simple structural examples is to reduce the influence of geometry and boundary conditions under well-defined stress - strain conditions. Examples of such tests are for instance; uniaxial compressive or tensile tests on concrete cylinders performed with the aim to verify that the obtained material behaviour corresponds with the given input. Other examples are for instance fracture tests of concrete beams subjected to three-point bending and tension stiffening tests in uniaxial tendons of reinforcing bars embedded in concrete members. Each type of test typically require a special material test and these material tests are in general described in codes for material testing, for instance those developed by RILEM.

One example of a basic material test that is recommended by the Dutch guideline for nonlinear numerical analyses of concrete structures, Rijkswaterstaat (2012), is given as an example here below and shown in Figure 10-1. The purpose of this test, is to study the interaction between damages caused by tension and compression respectively. The compressive strength of concrete specimens is reduced due to induced cracks that are parallel to the direction of the compressive stress.

In the first calculation step, a successively increased displacement $u_y(1)$ is defined in the vertical direction (y) to result in tensile stresses with the intention to cause a fully developed macro-crack in the concrete specimen. At the same time, a displacement $u_x(1)$ is defined in the horizontal direction (x) to give a compressive stress that is perpendicular to the "tensile load". The size of this "compressive load" is defined to correspond the to the poisons effect, i.e. $u_x(1) = \nu\, u_y(1)$.

After the concrete specimen has fully cracked, then a second calculation step is performed where the horizontal compressive displacement $u_x(2)$ in increased successively until a crushing failure occurs. The FE-analysist should in the first step show that the material model can give the uniaxial tensile curve that is given as input. In the second step, it can be seen how the material model accounts for reduction in compressive due to cracks. In Figure 10-2, an example of results from the benchmark example where it is clear that a reduction of the compressive strength is obtained as a result of lateral cracking. In the left figure, compressive stress-strain curves are given for varying ratios of tensile and compressive strain.

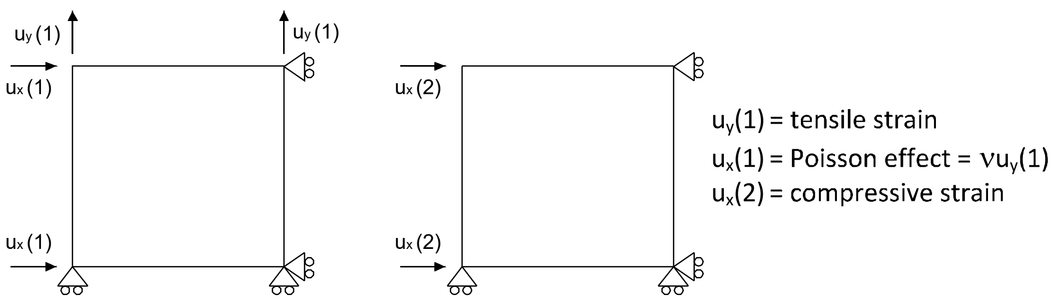


Figure 10-1 Example of a basic material test, from Rijkswaterstaat (2012).

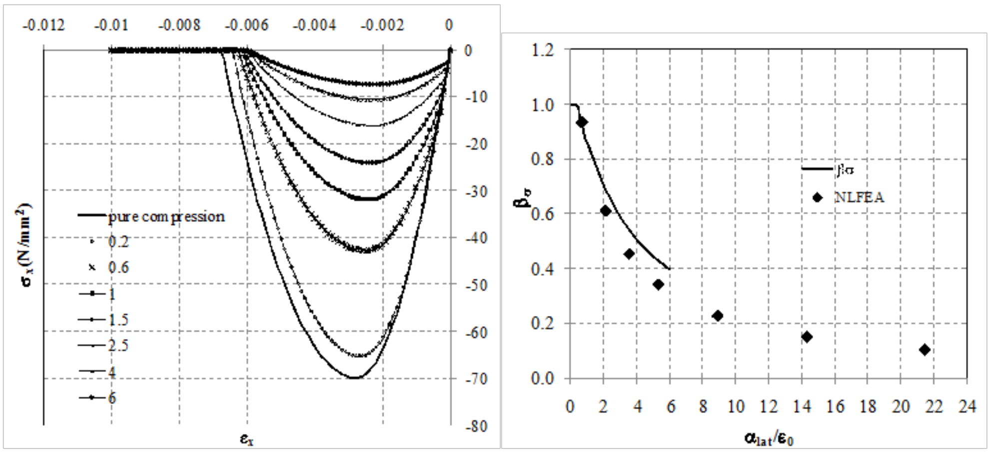


Figure 10-2 Example of results from the benchmark example, from Rijkswaterstaat (2012).

10.1.2 Structural test

The purpose of the structural test, is to validate that a certain structural response can accurately be reproduced by an algorithm or software. This is normally performed with a benchmark example. The intention of the benchmark example is to choose an adequate material model and software for a given structure or structural problem. It is important that the structural test is performed on the same type of structure as the structure that is going to be analysed. (Model Code, 2010)

The committee on "Computational Aspects of Analysis & Design of Dams" within ICOLD has over the last 25 years performed benchmark workshops (these are given every second year).

1. Bergamo, Italy, 1991
2. Bergamo, Italy, 1992
3. Paris, France, 1994
4. Madrid, Spain, 1996
5. Denver, USA, 1999
6. Salzburg, Austria, 2001.
7. Bucharest, Romania, 2003
8. Wuhan, China, 2005
9. St. Petersburg, Russia, 2007
10. Paris, France, 2009
11. Valencia, Spain, 2011
12. Graz, Austria, 2013
13. Lausanne, Switzerland, 2015

The ICOLD benchmark examples consist of generalized engineering problems that are devoted to bridge the gap between numerical analyses, the interpretation of results and their theoretical as well as practical relevance. These are therefore well suited for validating models in structural tests and to learn best practices from the dam community. The documentation from these benchmark workshops are available to the dam community through the internet and proceedings.

10.1.3 Mesh sensitivity

Mesh sensitivity tests should always be performed and preferably also presented in the project report. The purpose of these tests is to validate that the finite element mesh of the numerical model is sufficiently detailed. In Section 4.2, some recommendations regarding suitable element size was given. Based on experience, these are often rather good estimates of the mesh size that is required but it is up to the FE-analysist to ensure and show that the chosen element size is sufficient for the specific project.

In mesh sensitivity tests, at least three different discretisation's should be performed and if the results converge (coincide) with finer discretisation's then the mesh is sufficient. In linear elastic analyses, displacements and strain or stress should be compared in order to determine that a sufficiently small element size is used. In nonlinear analyses, the influence of different discretisation's on the structural response/resistance should be evaluated.

In cases with significant mesh sensitivity, where the results from two analyses with different element sizes does not show sufficiently similar results, then the results must be considered as not objective. (Model Code, 2010)

10.1.4 Verification analyses

In all projects, verification analyses should be performed. In cases with a simple structure and load case, which is relatively easy to analyse with means of traditional analytical calculations, then comparative calculations should be performed to ensure that the result from the numerical analysis is reasonable. This is in most cases possible to perform in order to assess if the numerical analysis gives reasonable results.

Concrete dams can often be idealized by simple structural components, for instance; arch dams (plane view arch and/or vertical columns), gravity dams (vertical columns), front-plates on buttress dams (slabs) etc., in order to obtain a rough estimation of the expected behaviour.

In projects with significant consequences with a complicated structural behaviour or load cases where it is not possible to obtain analytical results with reasonable accuracy, then an independent (simplified) numerical analysis should be performed. This can for instance be where a complicated 3D model of a dam is idealized as a simple 2D structure only considering the most important loads.

There are several different software's that may be used for performing these analyses. The analysis can for instance be performed in the same software as the detailed original analysis, except if a specific feature in this software is to be validated and no other possibilities to define this exists. This could for instance be if only one material model for nonlinear behaviour of concrete is available in the software. There is however always a risk to use the same software for verification analyses as the one used in the detailed analyses, since if there is a bug in the software that results in the wrong results than it is likely that this will affect both types of analyses.

The idealized simplified calculations may for instance also be performed with easier to use software's developed for the specific purpose of performing different types of analyses. Several different freeware software's exists that could be used for this. Examples of software's that can be used are for instance;

- CADAM – software for stability analyses of gravity dams and/or seismic evaluation. Developed by Leclerc et al. (2003)
  - Download (available at 2015-12-22): http://www.polymtl.ca/structures/doc/cadam/CadamCD.zip
- Response 2000 – software for nonlinear analysis of the cross-sectional load capacity of concrete structures. Developed by Bentz (2000)
  - Download (available at 2015-12-22): http://www.ecf.utoronto.ca/~bentz/r2k.htm
- HACON – software for analysis of temperature and stress in hardening concrete. Developed by Dahlblom and Lindemann (2000).
  - Download (available at 2015-12-22): http://www.byggmek.lth.se/english/resources/software/hacon-a-program-for-simulation-of-hardening-concrete/

Many other software's also exists. It should be noted, however, that freeware and other similar software's, requires higher degree of validation than commercial software's. The reason for this is that features in commercial software's are validated continuously. For instance, all commercial numerical software's have performed patch-tests, material tests etc., and the results from these are given in the manuals for the specific software. As described previously, the FE-analysist must always validate and verify the results numerical analyses, even if these are obtained from commercial software's. The reason is that significant errors in input, assumptions or wrong interpretation of the function of a features in a software may result in significant errors.

10.2 EQUILIBRIUM

In linear numerical analyses, it should always be checked that equilibrium is obtained, i.e. the sum of all reaction forces should be equal to the external loads. This check is in general performed (for energy quantities) in numerical FE codes. However some this check is not performed in quasi-static analyses performed with explicit solvers and some software's allows the analysis to continue even though if the tolerances are not met.

- If several loads are applied to the structure, then calculations should be performed so that the response from each load can be analysed and studied separately and that the superposition of these load effect is the same as the results obtained from the analysis with all loads considered. (Note: Superposition is not possible in nonlinear analyses)
- In nonlinear analyses, equilibrium is per definition never achieved. Instead, an iteration is considered to have converged if for instance the difference between external and internal forces is less than a pre-defined tolerance value. In some software's, results may be presented despite the fact that the increment has not converged for instance if the maximum number of pre-defined iterations has

been performed. Therefore, nonlinear analyses should be evaluated thoroughly regarding this to ensure that unbalanced forces and/or energy densities are small.

- In quasi-static or dynamic analyses based on an explicit solver, tolerances are not even defined. In these analyses very small load or time increments are used instead (typically 10^{-6} for concrete structures). Thereby, all analyses based on explicit solvers require much more extensive verification than other nonlinear analyses to ensure that a stabile solution has been obtained with sufficient accuracy. In quasi-static based on explicit solvers it should also ensured that the kinetic energy is negligible on both global and local level.

11 Safety format for numerical analyses

All material properties, loads and even models are associated with uncertainties. In an assessment or in design of a structure, it is important that these uncertainties are accounted for in a proper manner.

In Figure 11-1, an example of expected distribution of a concrete with mean values of tensile strength and elastic modulus of 2.5 MPa and 25 GPa respectively. According to probabilistic model code, JCSS (2001), both tensile strength and elastic modulus are considered have lognormal distributions with a covariance of 0.3 and 0.15 respectively.

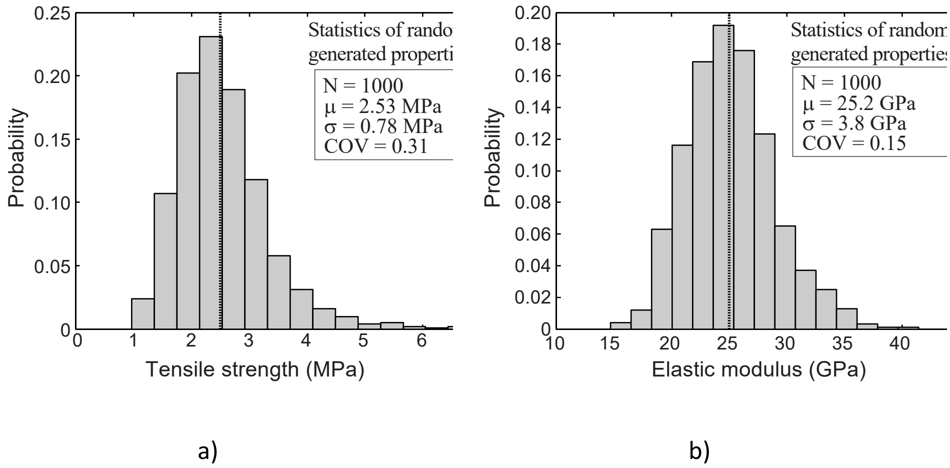


Figure 11-1 Example of a lognormal distribution of tensile strength and elastic modulus, from Malm et al (2011).

11.1 STRUCTURAL SAFETY CONCEPTS

The safety concept based on a factor of safety, sf (with the resistance R and the action S shall fulfil the following requirement:

$$S \leq \frac{R}{sf}$$

The factor of safety is selected on the basis of experimental observations, past experience, economic and political considerations and should be large enough to provide sufficient safety towards unwanted events that is assumed to occur if

$$S > R$$

The partial factor method, which is used in many design codes, is a development of the factor of safety implying that the permissible actions (or required resistance) shall satisfy the following relation

$$\frac{R}{\gamma_R} \geq \sum_{i=1}^{n} \gamma_{Di} S_{Di} + \sum_{i=1}^{m} \gamma_{Li} S_{Li}$$

where, S_{Di} is the dead loads, S_{Li} are the live loads and $\gamma_R, \gamma_{Di}, \gamma_{Li}$ are partial factors. Actions with high variability can with this format be given greater partial factors than those with low variability, and thus better representation of the uncertainties associated with actions and resistance is obtained. (Westberg 2010)

11.2 LINEAR ELASTIC FE ANALYSES

The safety concept is not an issue for linear analyses, since it is up to the FE-analysist to check the obtained results (through post-processing) according to the used design code or guideline. Thereby, in linear numerical analyses, the design values of loads can be defined directly and stress, sectional forces etc. are obtained from the FE-model and compared to the allowed value.

If the partial coefficient method in Eurocode is used, then all loads are defined with their design values and the linear elastic material properties are defined as their design values. The FE-analysist can in the post-processor ensure that for instance maximum principal stresses are lower than the design value of the tensile strength or that the minimum principal stress does not exceed the design value for the compressive strength. In many codes, it is defined that the compressive stresses must be lower than a factor of the compressive stresses in order for linear creep to be considered. In Eurocode 2, this is defined as 45 % of the characteristic cylinder strength (f_{ck})

$$\sigma_{c.\max} = 0.45 f_{ck}$$

11.3 NONLINEAR FE ANALYSES

For assessment of the load capacity with nonlinear FE analyses and for assessment of displacements, cracks etc. in the serviceability state; a suitable safety format has to be chosen. The safety format should be considered the uncertainties in material properties, model uncertainties and uncertainties in the loads.

To verify that deformations and acceptable crack widths are within the acceptable limits in serviceability state, normally characteristic material properties are used. The determination of the tensile strength of the concrete should then also consider to internal stresses and/or cracks that might have occurred due to hydration in early age concrete or due to shrinkage. One common method to consider this is to reduce the characteristic tensile strength with a crack limitation factor, typically 2.

To assess the load capacity in ultimate limit state with the use of nonlinear FE analyses, normally one of the two following approaches are used

- Partial safety concept – where the design values are used for the strength of the material, i.e. the strength is reduced with partial coefficients that consider uncertainties due to material distribution and or model uncertainties.
- Global resistance methods – where mean values are used in the numerical analysis and the resulting load capacity is divided with a chosen global safety factor to obtain a design load capacity. The global safety factor is then a product of a factor for the global distribution in material properties and due to model uncertainties.

The latter method with a global safety factor is to prefer, since it gives a more realistic analysis of the potential failure mode that will result in failure. For instance, in design of a concrete beam the first method may for instance indicate that a shear failure is the failure mode while the analysis with global safety concept indicates a bending failure. In these cases, two different kinds of reinforcement solutions are required to increase the strength of the beam where the shear reinforcement does not contribute to increasing the strength due to bending which is the most likely failure mode. More information regarding this method is found in Concrete report 15 from the Swedish Concrete Association and fib Model Code (2010).

Another method that could be used as a safety concept in numerical analyses is probabilistic analyses. In probabilistic analyses, the input parameters are given as probabilistic functions and Monte Carlo simulations are performed. Thereby significant amounts of analyses are required in these cases, and based on the outcome from these, a reliability index or probability of failure can be calculated. This approach is often not suitable to use in combination with numerical analyses due to the extremely long cpu-time that would be required.

11.3.1 Global resistance methods

In a nonlinear analysis, the response of the structure is best described if mean values are used for the material properties that are included in the analysis. All numerical material models and finite element programs are validated against experimental results based on average values. A global safety criteria adjusted to the partial factor method can be rewritten as

$$\frac{R_m}{\gamma_{Rg}} \geq \gamma_{S1} \cdot S_{k1} + \gamma_{S2} \cdot S_{k2} + \cdots + \gamma_{Sn} \cdot S_{kn}$$

where,

R_m is the load capacity obtained from a nonlinear analysis based on mean values of all material parameters

γ_{Rg} is a global factor of safety of the structural load capacity.

The global safety factor should consider the uncertainties for the variables that determine the load carrying capacity of the structure and the uncertainties of the numerical model. The safety is related to the average values of the load carrying capacity, and therefore γ_{Rg} represent a global central safety factor for the load capacity. The difficulty with this method is that the global safety factor incorporates a wide and unspecified number of uncertainties, according to Carlson et al. (2008). (Malm 2012)

In fib Model Code (2010), two different methods for obtaining a global resistance factor are presented, which are summarized below.

Global resistance factor method

In the global resistance factor method given in fib Model Code (2010), the design resistance of a structure can be calculated as

$$R_d = \frac{R_{m.adj}}{\gamma_R \cdot \gamma_{Rd}}$$

where,

R_d is the design resistance

$R_{m.adj}$ is the resistance obtained from the nonlinear analysis based on adjusted mean input material values as described below.

γ_R is the partial factor of resistance, equal to $\gamma_R = 1.2$

γ_{Rd} is the model uncertainty factor. For well validated numerical models, it is proposed by fib Model Code (2010) to define the model uncertainty equal to $\gamma_{Rd} =$ 1.06. For low-level validation, higher uncertainty values should be used. In chapter 4.6.2.2 in fib Model Code (2010) it is recommended that $\gamma_{Rd} = 1.1$ is used for models with high uncertainties. It should be noted that this factor does not cover errors due to approximations in the numerical models.

The nonlinear FE analysis should be performed with adjusted mean values for the concrete and steel. The background for this is to adjust for different partial safety factors (1.15 for steel and 1.5 for concrete). In addition, the concrete parameters are, in addition, reduced with a factor α_{cf} to account for sustained load effect and unfavorable effects of load application. The range of this parameter is $0.85 \leq \alpha_{cf} \leq$ 1.0 and according to fib Model Code (2010) the lower value should be used unless these effects can be considered as non-existent.

The reinforcing steel properties to be used are defined as

$$f_{ym} = 1.1 f_{yk}$$

where f_{ym} and f_{yk} are the mean and characteristic value for the yield stress of the steel.

The concrete properties to be used are defined as

$$f_{cmd} = 0.85 \cdot \alpha_{cf} \cdot f_{ck}$$

where f_{cmd} and f_{ck} are the adjusted mean and the characteristic value for the compressive strength of concrete. The parameter $\alpha_{cf} = 0.85$ according to the recommendations mentioned above.

Other concrete properties, such as tensile strength, fracture energy, bond strength, should all be reduced in the same way as the compressive strength.

Method of estimation of a coefficient of variation of resistance (ECOV)

It is not rational to use a general value of the global safety factor γ_{Rg} for all concrete structures regardless of the failure mode in question.

One general method described in Sustainable Bridges (2007) to determine the global safety of the structure is to perform two separate analyses of the load capacity of the structure. The first analysis should be based on the mean values of material properties while the second analysis should be based on characteristic material values. From these two analyses, the load capacity from the model based on mean values R_m and the load capacity R_k from the model based on characteristic material properties are obtained. An updated definition of this approach is given in Model Code (2010) and this approach is given below.

Based on the assumption that the distribution for the load capacity is lognormal, the coefficient of variation V_{Rf} can be determined as

$$V_{Rf} = \frac{1}{1.65} \ln\left(\frac{R_m}{R_k}\right)$$

The global safety factor γ_{Rg} can then be estimated based on the following equation

$$\gamma_{Rg} = \exp\left(\alpha_R \cdot \beta_T \cdot V_{Rf}\right)$$

where,

α_R is the sensitivity factor for R and can approximately be set equal to the conservative value $\alpha_R = 0.8$

β_T is the target value for the safety index according to design codes (see Section 2.4)

The design resistance can thereby be calculated as

$$R_d = \frac{R_m}{\gamma_{Rg} \cdot \gamma_{Rd}}$$

where,

γ_{Rd} is the model uncertainty factor (see the previous section).

12 Practical modelling tips

In this chapter, a few practical modelling tips are given that may come in handy when performing FE-analyses of hydraulic concrete structures (and in most cases, other concrete structures as well). These modelling tips given in this chapter are based on the authors experience over the years.

12.1 COMBINE PROGRAMMING WITH FE-SOFTWARE'S

The most powerful and practical modelling tip is by far to use programming to develop or enhance the numerical model, submitting multiple analyses or to extract data from FE-programs and to post-process results as graphs, etc.

Many FE-programs are based on text-based input codes, such as Abaqus, Ansys, Adina, Atena, Brigade Plus, Diana, Solvia etc. In addition, these programmes can also be run from the commando prompt (CMD). Thereby, for these FE-software's it is possible to generate part of the input codes with a programming software's and also submit the input file to the FE-solver from batch mode (i.e. the commando prompt).

Some of the software's graphical user-interface also have built-in programming languages (for instance python) and it is thereby fairly easy to use to enhance the FE-modelling.

Otherwise, software's such as for instance Matlab is something that the author has used a lot to create features in the model that needs a lot of repetition in the graphical user-interface. This could for instance be a script to generate individual node-to-node springs for all nodes in two surfaces.

Matlab commands

In this section, some examples on commands in Matlab that are useful when writing input codes. In this specific example, Matlab is used to modify an Abaqus input file from Abaqus CAE, but as mentioned previously this could just as easily be performed with any other software.

Use a dynamical variable for the name of the final input-file:

```
for n=1:10 %variable that is created in a for-loop
fnameMain =
sprintf('%s%d%s',NameInpFile_',n,'.inp');
fidMain = fopen(fnameMain, 'w');
```

In this case, ten separate input-files will be created with the names: NameInpFile_1.inp to NameInpFile_10.inp.

After this is just to fill the input file with code. Normally, the FE-analysist creates a model in the graphical user-interface of the FE-software (or creates the geometry from a third party software as described in Section 3.3) and exports it as an input code (in asci text). It is often recommended that this input code is split up into different parts (depending on what needs to be altered by coding) or at least be altered in a text-based software (such as Matlab or Notepad++), for instance remove the last lines that closes the input file.

For instance, if an input file based on a model have been created through the user-interface and the code is only used to perform several analyses where the load is changed. In this case, the modified input file from the graphical user-interface can be included

```
fprintf(fidMain,sprintf('%s', '*INCLUDE,
INPUT=Original_inpfile.inp \n'));
```

The command (`\n`) is needed to clarify that a line break is needed at the end of the input line. The Matlab coding that constitute the new loads can then be added afterwards and the analysis can be submitted to the FE-solver, for instance

```
jobn= fnameMain(1:end-4);
disp('Running ABAQUS')
dos(['abaqus job=' jobn ' cpus=4 interactive']);
```

Now the analysis progress will be seen in the commando prompt of Matlab and if all the code is written in a for-loop (as illustrated above) then the next model (NameInpFile_2.inp) will be submitted as soon as the first analysis has finished or aborted.

12.2 PUSH-OVER ANALYSES OF GLOBAL FAILURE MODES

Dam stability analyses are in general based on global safety factors for different failure modes such as sliding and overturning.

In RIDAS (2011), the safety factor for sliding (s_g) together with the friction angle (δ_g) defines the maximum limit of the friction coefficient (μ_{limit}). This factor should be higher than the coefficient of friction in the sliding plane ($\mu_{analytical}$); expressed as the ratio of horizontal forces ($\sum H$) divided by the vertical forces ($\sum V$) in the sliding plane.

$$\mu_{analytical} = \frac{\sum H}{\sum V} \leq \mu_{limit} = \frac{\tan \delta_g}{s_g}$$

Rewritten, the safety factor can be expressed as

$$s_g = \tan \delta_g \cdot \frac{\sum V}{\sum H}$$

The safety factor regarding overturning is in RIDAS (2011) defined as the ratio of stabilizing moments ($\sum M_R$) divided by overturning moments ($\sum M_S$).

$$s_{analytical} = \frac{\sum M_R}{\sum M_S}$$

The safety factor for a sliding failure can easily be obtained from a static analysis of the design load case. However, in order to obtain the factor of safety for overturning it is required that an analysis of the actual failure is performed. When performing a FE-analysis it will only be possible to obtain the failure of the dam (and thereby the safety factor) if a progressive push-over analysis or a reduced strength analysis is performed.

However, depending on how the loads are increased or the strength reduced, different safety factors will be obtained. It could for instance be one safety factor for each load etc. This was for instance shown in the NW-IALAD project, NW-IALAD (2006). In addition, it is likely that the failure of the dam is due to combination of failure modes. Analyses have shown that one common failure mode is that due to the overturning moments tensile stresses occurs in the upstream toe of the dam which results in that a sliding failure may initiate even for loads lower than obtained from the analytical calculations, see for instance; Malm et al (2016a), Malm et al (2016b). Another failure mode that can occur is limit overturning due to crushing of rock or concrete in the downstream toe, Fishman (2007), Fishman (2009), Broberg and Thorwid (2015). Another example of a failure mode could be due to cracking of the dam which could result in internal failure modes, see Malm et al (2016b), Fu and Hafliðason (2015), Broberg and Thorwid (2015).

It is important that the safety format used is based on the same principles as the traditional analytical calculations, in order for the FE-analyses to be comparable to analytical calculations and for other practicing dam engineers to be able to interpret the safety factor.

The intention of this section is to present an approach that results in similar safety concept as the analytical failure analyses. The principle is that the analysis is performed in at least two steps

1. Normal loads
2. Progressively increased overturning loads

In analyses with frictional contact, it is often necessary to perform an initial step with only the gravity load. After this step, all other normal loads may be applied. The reason why the gravity load has to be applied first, is to ensure that the frictional resistance is sufficient for the step with normal loads.

At the end of the step with the normal loads, the normal (often vertical) and tangential (often horizontal) forces may be extracted for all loads in the sliding plane. After this, the frictional coefficient can be calculated.

In the step with the progressively increasing loads, only the overturning forces (or force components) that are increased successively. All overturning forces should increase with the same load factor and when the failure occurs in the FE-model the last load increment corresponds to the obtained safety factor of the dam.

The loads that normally should be increased are

- Uplift pressure
- Horizontal water pressure (upstream only)
- Ice load

Other loads that are stabilizing the dam, such as gravity, vertical water pressure etc., should maintain constant at their normal load level throughout the progressive failure analysis.

In addition to this, in order for the numerical analyses to be comparative to the analytical calculations, it is important that it is only the load values that increase (i.e. the resultant force) and the position of the level arm for each force should be maintained constant. This procedure is analogue with increasing the density of the water instead of simulating an overtopping scenario. The increased density approach is illustrated in Figure 12-1.

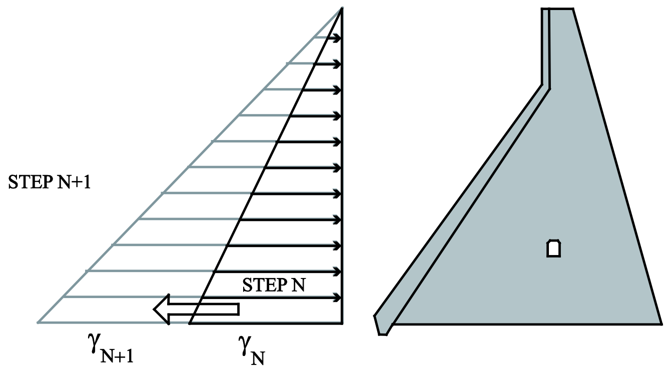


Figure 12-1 Principle to increase the horizontal water pressure until the numerical analysis reaches failure.

By using this approach, identical results with an analytical calculation will be obtained (if a pure sliding or overturning failure is simulated), as long as the dam behaves as a rigid monolith during the progressive loading scenario. As mentioned previously, the obtained failure mode in the numerical analysis is always the potential failure mode that has lowest factor of safety and it is rather common that the obtained failure is a combination of different modes.

If for instance a pure overturning failure should be simulated, then the dam should not be able to slide due to slipping. This can for instance be performed with horizontal linear springs that are defined between the dam and the foundation perhaps in combination with nonlinear springs in the vertical direction that only can transmit compressive forces. Thereby, it will still be possible to calculate the coefficient of friction and thereby the risk of sliding without actually simulating the sliding failure. The nonlinear vertical springs still allows an overturning failure to occur.

12.3 NONLINEAR ANALYSES WITH MATERIAL NONLINEARITY

Performing nonlinear analyses with material nonlinearity is much more time consuming than performing analyses based on linear elastic behaviour.

Some sources of instability include local yield, snap-through, surface wrinkling or localised material failure. When local instabilities develop, it may not be possible to obtain a static solution. At a brittle failure a crack (often diagonal shear crack) usually initiates rapidly, where it's nearly invisible just before the peak load, and after the peak load it dominates the appearance. (Malm 2009)

Modelling this as a static analysis, will in most cases result in convergence difficulties due to local instabilities. In general, this result either in that the analysis aborts early in the cracking process, when a cracked element need to unload to its surroundings or just when the peak response is reached.

In general the more complicated the material model is, the more difficult to achieve convergence. As mentioned in Section 5.3 many different material models exists and as a rule of thumb it can be said that isotropic damage models are the least sensitive while plasticity based models are more difficult.

12.3.1 Solution methods and loading procedure

If the analysis is performed as load controlled, i.e. with a load that successively needs to increase for each increment, it is not possible to capture the response if unloading is necessary, as illustrated in Figure 12-2. Thereby, if the unloading part is to be captured, a different loading/solution procedure is required. Normally when solving analyses with material nonlinearity, the full Newton Raphson method is used. However if the unloading part of the curve is to be capturedm a different solution method is required, such as the Arch length method which allows for decreasing load increments.

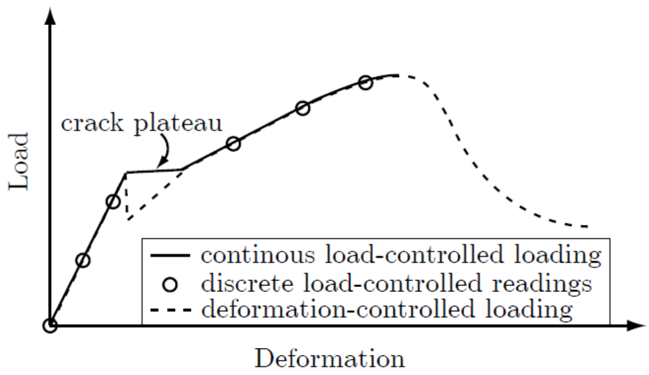


Figure 12-2 Difference in obtained response depending on if load-controlled or a deformation-controlled loading procedure is used. From Malm (2006).

Otherwise, one effective approach is to replace a prescribed load with a prescribed displacement instead. In these cases, the corresponding force can be obtained as reaction force in the nodes where a deformation was prescribed. This is easy to do in cases where for instance a point load is applied to the structure. However, for more complicated load cases, for instance in a global stability analysis of a dam, it is not so easy to replace the loads with prescribed deformations.

12.3.2 Avoiding convergence difficulties

There are several ways to some extent avoid these convergence difficulties. Different approaches that the FE-analysist should consider are described in the following. Suggestions regarding how to avoid convergence difficulties in Abaqus, is also presented by Malm (2006) and Malm (2009).

Refine the model to remove numerical singularities

The most obvious method is to define the mesh, boundary conditions and loads to minimise potential numerical singularities. If convergence issues occur in the model, it is possible in the post-processor to study which elements that are unable to converge. Redefining the mesh in these regions may in most cases help the analysis go at least a bit further.

Change tolerances and number of allowable iterations

It is possible to change tolerances in order to force the analysis go further. It is not recommended to increase the tolerances, i.e. allowing larger errors for each iteration. The reason for this is that these errors will likely result in secondary errors. Therefore, it is much better to define stricter tolerances (allowing smaller errors than default) but instead increase the amount of iterations that should be performed before convergence is checked. In addition, it should also be defined that smaller load increments (than default) may be used and allowing for a larger number of iterations before convergence is checked.

In some software's, regularization of the constitutive equations may be allowed in the material models. This is a sort of tolerances where it allows that stresses exceeds the values given by the failure surface. To the authors experience, using regularization is a fairly good way to improve convergence without compromising the results. This, as long as the regularization is very small.

Using suitable elements

In nonlinear analyses, rather small elements are required as described earlier in Section 4.2.2. To the author's opinion, it is in general easier to perform nonlinear analyses with a fine mesh consisting of lower order elements with reduced integration compared to using higher order elements. The downside with these types of lower order elements is hourglass deformations, as described in Section 4.1.3. The hourglass deformations results in zero strain in the integration points and hence these elements are often the reason for not finding convergence. One efficient way to overcome this, which is available in some software's, is that an hourglass control is defined where artificial energy is introduced to prevent the hourglass deformations.

It is also a good idea to test with different type of elements for instance hexahedral vs. tetrahedrons.

Introduce damping

One common reason for not finding convergence is that the amount of energy being released due to cracking, which results in that the energy tolerances are exceed. One way to reduce the effect of this, is to introduce numerical damping in the model. This can be done by introducing discrete dashpots to all nodes in all directions or by using built-in functions that allows for automatic stabilization.

Perform quasi-static analysis

If possible, the problem might be converted into a quasi-static analysis where an explicit solution scheme is used. Point loads and distributed loads can be converted to velocities of a small magnitude or displacements with a slow increasing amplitude initially. Since an explicit solver is used (which is primarily intended for dynamic analyses), it is important to ensure that there is only marginal kinetic energy in the model. It should be ensured that only small amount of kinematic energy occurs, compared to the strain energy. This should be checked both on a global level for the whole model, but also on element level so that no element is subjected to high kinetic energy. This is important since one single small element subjected to high kinetic energy will not affect the energy of the entire model, but may have significant influence on the structural response.

On a global scale, the kinetic energy should, as a rule of thumb, be less than a few percent. The exact amount depends on the specific model and when in the loading procedure, the dynamic response occurs. The largest peaks in kinetic energy is often obtained directly after a crack has initiated. For some models, it might be possible to allow for up to 10 % kinetic energy compared to strain energy, while in others everything above 1 % will influence the results significantly.

An explicit solver marches a solution forward through time in small time increments without solving a coupled system of equations at each increment (or even forming a global stiffness matrix). Each increment is fast and easy to solve but very small time increments are used (usually of the magnitude 10^{-6} s). Thereby, in order to obtain reasonable cpu-times it is important to perform as short time steps as possible or use mass scaling in order to increase size of the stable time increment.

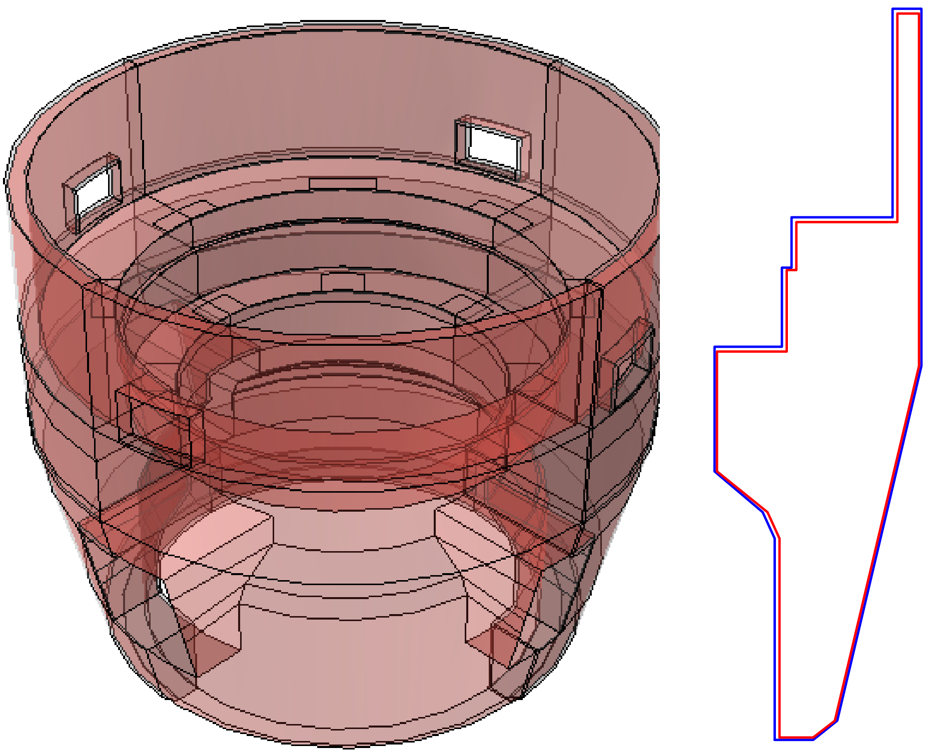
12.4 MODELLING OF REINFORCEMENT LAYERS

Modelling of reinforcement can be a time consuming task, and depending on the level of detail required from the analysis, different approaches may be used.

In many cases, it is sufficient to consider the effect from the reinforcement as smeared over the structure. In some software’s it is possible to define smeared reinforcement directly for solid elements. In these cases, the reinforcement area of one bar, its direction, bar spacing and placement in the cross-section are required input.

In some software’s, this is not possible to define smeared reinforcements for solid elements. In these cases, the following approach may be used instead.

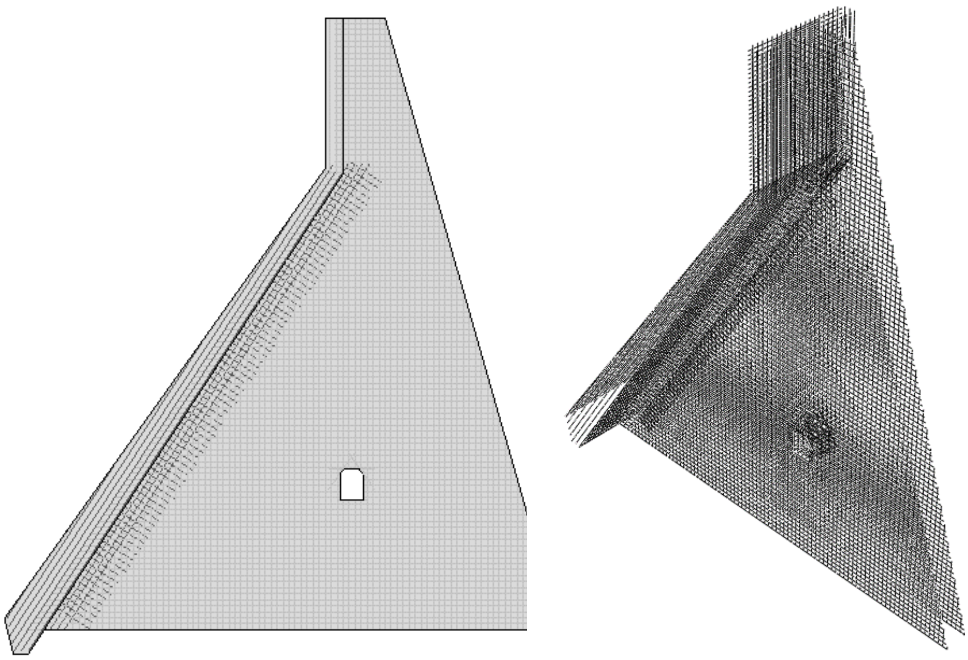
One approach to include the reinforcement in this manner is to create a part consisting of membrane (or shell) elements. This part should be defined with an offset from the concrete surface, corresponding to the concrete cover. The membrane elements are defined with a dummy material properties (or concrete material properties) and with an insignificant thickness, for instance 10^{-6} m. Thereby the membrane part should have no direct effect of the model. This part is only used as modelling aid in the FE-model. Most software's allows for definition of reinforcement layers on shell or membrane elements. Thereby, in addition to the dummy material properties, reinforcement layers with the actual properties and characteristics of the reinforcement is given. The membrane elements may also be divided into different areas where different amount of reinforcement is given in these areas. A model where this approach has been used is illustrated in Figure 12-3.



a) b)

Figure 12-3 Example of membrane elements (illustrated with red colour) embedded within the concrete volume.

If high level of detail is required, where the results needs to be extracted for specific reinforcement bars at specific locations, then the reinforcement bars should be modelled individually. In this case, all reinforcement bars are modelled as individual beam or truss elements. An example of this is shown in Figure 12-4.



a) b)

Figure 12-4 Example where the reinforcement has been modelled as separate bars.

12.5 EXTRACTING SECTIONAL FORCES

As mentioned previously in Section 3.1, concrete dams are most often modelled with solid elements in 2D or 3D. It is not possible to obtain sectional forces from solid elements as it is for shell or beam elements.

However, sectional design according to codes (such as Eurocode), is based on sectional forces (moments, shear and normal forces). Therefore, in order to use the results from the FE-analysis for designing reinforcement etc. it is necessary that sectional forces can be extracted.

One approach is to integrate stresses over the cross-sectional thickness. This is however, not a suitable approach to use in design purposes since it often cumbersome to do this integration since the FE-analysist has to manually define the sections that should be integrated.

One other approach that, in the authors opinion, is much more effective is to embed shell elements within the solid elements as modelling aid. The shell elements should be defined to describe the geometry of the solid element structure as far as possible. Thereby, it should have the same geometry as the solid structure but defined in and constrained to the mid-plane of each cross-section. The thickness of the shell elements should be equal to the thickness of the solid structure.

It is important that the shell elements are defined with a low stiffness and mass so that it does not affect the structural response of the original solid element structure. Thereby, all shell elements should be defined with a dummy value of the elastic modulus, typically 1 [Pa].

Sectional forces will, due to this, be possible to obtain from the shell elements, but since the shell elements are defined with an elastic modulus equal to 1 Pa, then all sectional forces must be multiplied with the elastic modulus of concrete. An example of an intake structure where this technique has been used is illustrated in Figure 12-5.

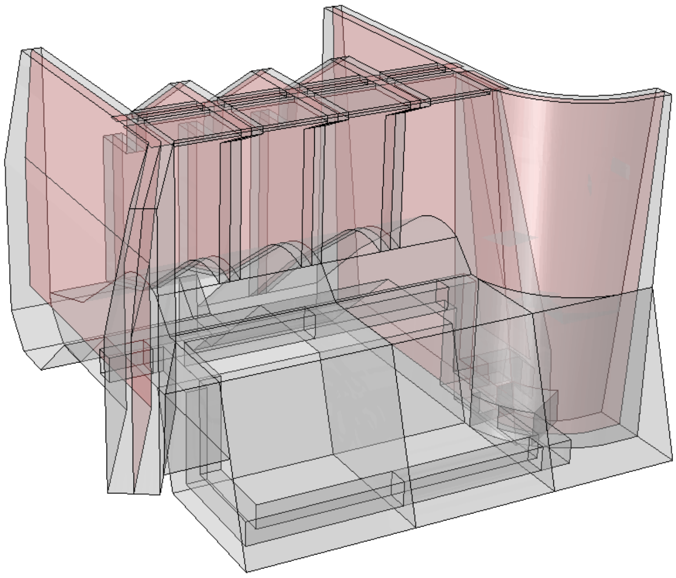


Figure 12-5 Embedded shell elements (illustrated with red colour) within the 3D solid intake structure. The embedded elements are only used to extract sectional forces.

12.6 PRE-STRESSED TENDONS

12.6.1 Pre-stressing and seating of tendons

The pre-stress in tendons can often be defined as an initial stress and/or as an equivalent temperature. If the stress is included as an initial stress, this corresponds to a pre-tensioned tendon. The reason is that the pre-stress will be allowed to change during an equilibrating static analysis step.

Pre-tensioned tendons are, however, rarely used for hydropower applications, it is mainly post-tensioned cables that are used especially to increase the dam stability.

Therefore, in order to model post-tensioned tendons, the pre-stress must be kept constant during the initial equilibrium solution. Some software's gives this possibility.

In some software's it is possible to define special types of elements that you can predefine a given force or relative displacement of. With these elements, it is possible to define the pre-stressing force and the tendon slip due to seating, i.e. locking the tendon into place. This may be done so that one end of the special type of element is connected to the active end of the tendon, while the other end is connected to a reference node in space. This reference node is then connected to an anchor plate through a multi-point-constraint so that the tendon force is transmitted into the concrete structure. An example of this procedure is shown in Figure 12-6, where the special type element used here is named connector elements in Abaqus.

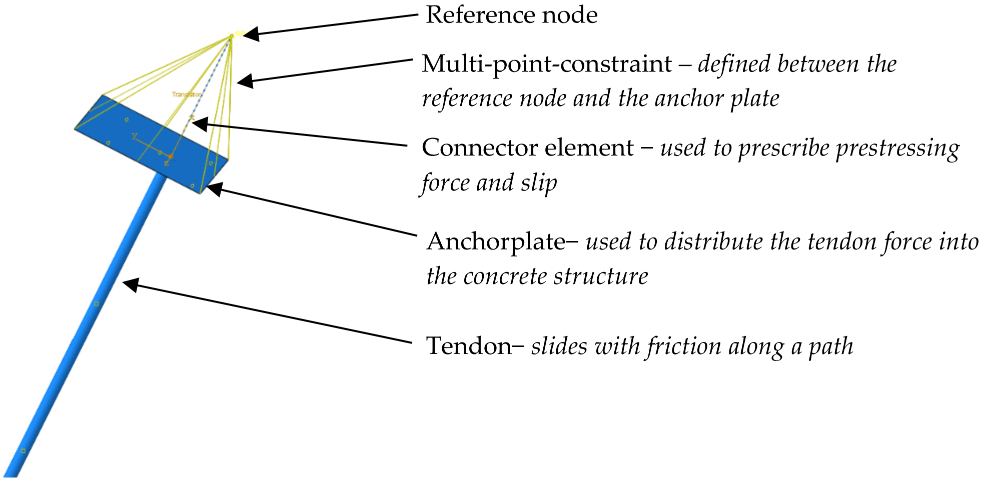


Figure 12-6 Example of how to simulate the pre-stressing procedure.

12.6.2 Interaction between tendon and concrete

Grouted tendons

Grouted pre-stressed tendons can in most software's be considered rather simply by defining truss or beam elements representing the tendon, which is connected to the concrete structure as an embedded constraint. The embedded constraint is basically constructing rigid links where the nodes of the tendon are constrained to the closest nodes in the concrete.

Unbonded tendons

If ungrouted tendons, i.e. tendons that can move freely inside ducts are to be modelled this is in general more complicated. A very simple approach may be to replace the tendon with a compressive force subjected to the dam at the location of the anchor. This procedure may be sufficient for straight tendons but if the tendons are curved, the radial force also has to be considered.

In order to consider ungrouted tendons in a more detailed manner, several different approaches may be used to simulate this. The general procedure is however that the tendon has to be able to slide (with friction interaction) along a pre-defined path, i.e. the duct, and the duct is constrained to the concrete structure (in similar manner as described for the grouted tendon).

One way to model this could be with friction truss elements, as described in Dameron et al (2003). In this setup, the friction truss ties are defined with an angle that corresponds with the friction angle. One downside is however, that seating (i.e. the tendon slip that occurs at locking the cable into place) requires backward friction. Therefore, the friction truss ties have to be directed in the opposite direction for the length that is subjected to the slip during seating, as seen in Figure 12-7.

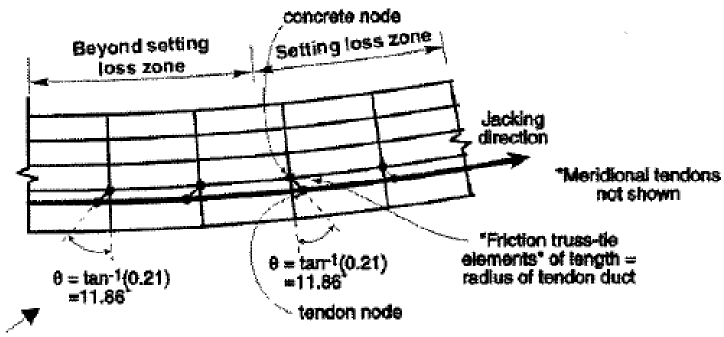


Figure 12-7 Example of friction truss ties that are used to describe friction along an unbonded tendon, from Dameron et al (2003).

Different other approaches to model unbonded tendons are for instance described and discussed in the ISP 48 project report, NEA (2005). One approach that may be simple to define in most software's is that a shell element is embedded in the concrete structure (i.e. fully restrained), where the shell element defines the path of the tendon. The tendon (beam or truss element) is then connected to the shell with friction in the tangential direction and does not allow for separation in the normal direction. Thereby the tendon can move freely along the duct (shell) and describe losses due to friction.

In some software's, special purpose elements are available that can be used to describe a sliding contact between beam (or truss) elements. These are for instance tube-to-tube elements in Abaqus or T-Beamstress in Solvia. In these cases, one outer tube is defined corresponding to the tendon duct, which is constrained to the concrete volume and an inner beam (or truss) element which is connected to the outer tube with for instance friction.

12.7 ESTIMATE CRACK WIDTHS

In some software's, it is possible to calculate the crack width directly from a material model based on smeared crack approach. If this is not possible in the software used, then the FE-analysist can perform their own calculations of crack width as described in this section.

Calculating the crack width can be performed in different ways. In all models, it is possible to calculate the crack width by integrating the total crack opening displacement within a predefined distance, i.e. by multiplying the total crack opening displacement within the crack band with the length of the crack band. The crack band length was described previously in Section 5.1.2. The crack width can thereby be calculated, as described in Cervenka et al. (2015) and Malm (2009), as follows

$$w_{cr} = \varepsilon_{cr} \cdot L'$$

where,

w_{cr} is the crack opening displacement [m]

ε_{cr} is the cracking strain

L' is the length of the crack band in the direction perpendicular to the crack direction [m]

This procedure is done to reduce the effect of mesh sensitivity, both for element size and element orientation for skew meshes. The length L' can be calculated as follows

$$L' = \gamma L$$

with

$$\gamma = 1 + (\gamma_{max} - 1) \cdot \frac{\theta}{45°},\ 0° \leq \theta \leq 45°$$

where,

θ is the minimum of the angles θ_1 and θ_2 between the direction normal to the failure plane and the element sides. [°]

L' L is the length of the crack band with skew mesh [m]

γ_{max} is a factor accounting for skew mesh, the recommended value according to Cervenka et al. (2016) is $\gamma_{max} = 1.5$

Malm and Holmgren (2008) and Malm (2009), showed that this method gives results that corresponds well with measured crack widths in laboratory tests and observed crack widths on real structures.

If the concrete dam is reinforced, it is also possible to calculate the crack width by extracting the strain in the reinforcement at the location of the crack and use this strain in the equations in for instance Eurocode 2 for crack widths.

12.8 SUBSTRUCTURE MODELLING

One feature that is becoming more and more used in FE analyses of complex structures consisting of different parts, and especially in cases with multiple use of parts etc. is called substructure modelling. This approach may for instance be used where structural components with a well-defined structural function are grouped together resulting in that their internal degrees of freedom may be eliminated from the global analysis. Using a substructure makes the model definition easier and the analysis faster when the global structure consists of at least some structural component is used several times. This could for instance be whole monoliths of a dam, or for instance lateral bracing beams between supporting walls in buttress dams etc.

In these cases two separate analyses are performed. In the first analysis, only the substructure is analysed, where the nodes that are constrained to the other parts of the structure or subjected to boundary conditions are retained. In the second analysis, of the whole global system, the substructures are connected to the rest of the model by the retained degrees of freedom at the retained nodes.

13 Acknowledgement

The work presented in this report was carried out as a part of "Swedish Hydropower Centre - SVC". SVC has been established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, KTH Royal Institute of Technology, Chalmers University of Technology and Uppsala University.

Participating companies and industry associations are: Alstom Hydro Sweden, Andritz Hydro, E.ON Vattenkraft Sverige, Falu Energi & Vatten, Fortum Generation, Holmen Energi, Jämtkraft, Jönköping Energi, Karlstads Energi, Mälarenergi, Norconsult, Skellefteå Kraft, Sollefteåforsens, Statkraft Sverige, Sweco Energuide, Sweco Infrastructure, SveMin, Umeå Energi, Vattenfall Research and Development, Vattenfall Vattenkraft, Voith Hydro, WSP Sverige and ÅF Industry.

This report is a result of experiences gained from different research and consulting projects. Many people have thereby contributed to this report, in some way, and to those am I truly grateful. I especially want thank Rikard Hellgren (KTH/WSP), Tobias Gasch (KTH/Vattenfall) and Daniel Eriksson (KTH/SWECO) who has contributed with suggestions and input to this specific guideline and have also performed the review of this report.

In addition, I want to thank the reference group of SVC structural engineering that have contributed with review, ideas and suggestions for this report. The reference group of SVC consists of; Anders Ansell (KTH), Carl-Oscar Nilsson (Uniper), Daniel Eriksson (KTH/SWECO), Erik Nordström (KTH/SWECO), Fredrik Johansson (KTH/SWECO), Håkan Bond (WSP), Lars-Elof Bryne (Vattenfall), Magnus Svensson (Fortum), Manouchehr Hassanzadeh (SWECO/LTH), Marcus Hautakoski (VRF), Marie Westberg-Wilde (KTH/ÅF), Martin Hansson (Statkraft), Martin Rosenqvist (LTH/Vattenfall), Mats Persson (Vattenfall), Mats Stenmark (Norconsult), Rikard Hellgren (KTH/WSP), Robert Lundström (Skellefteå kraft), Tobias Gasch (KTH/Vattenfall) and Tomas Ekström (ÅF).

14 References

Andersson O & Seppälä M (2015): Verification of the response of a concrete arch dam subjected to seasonal temperature variations, MSc Thesis, TRITA-BKN-Examensarbete 458, KTH Royal Institute of Technology.

Aswegan K & Charney F.A (2014): A simple linear response history analysis procedure for building codes. In: Tenth U.S. National Conference on Earthquake Engineering Frontiers of Earthquake Engineering, July 21-25, Anchorage.

Attarnejad R & Lohrasb, A (2008): Reservoir length effect in calculation accurate of dam-reservoir interaction. In: The 14:th World Conference on Earthquake Engineering, 12-17 October, Beijing, China.

Bangash M (2001): Manuals of numerical methods in concrete – Modelling and applications validated by experimental and site-monitoring data. Thomas Telford, ASCE Press, Reston.

Bazant Z.P, Hauggaard A, Baweja S & Ulm F (1997): Microprestress-Solidification Thory for Concrete Creep. 1: Aging and drying effects. Journal of Engineering Mechanics, Vol. 123, No. 11, pp 1188-1194.

Belytschko T & Black T (1999): Elastic crack growth in finite elements with minimal remeshing. International journal for numerical methods in engineering, Vol. 45, No. 5, pp 601-620.

Bentz, E.C (2000): Sectional Analysis of Reinforced Concrete Members. PhD Thesis, Department of Civil Engineering, University of Toronto.

Björnström J, Ekström T & Hassanzadeh M (2006): Cracked concrete dams – overview and calculation methods, Report 06:29, Elforsk AB, Stockholm, Sweden, 2006 (in Swedish).

BKR (2010): Regelsamling för konstruktion BKR 2010. Boverket, Karlskrona.

Blomdahl J, Malm R, Nordström E & Hassanzadeh M (2016): Minimiarmering i vattenkraftens betongkonstruktioner. Energiforsk report 2016:234, Energiforsk, Stockholm.

Broberg L & Thorwid M. (2015): Evaluation of Failure Modes for Concrete Dams. 2015. MSc Thesis, TRITA-BKN-Examensarbete, 455. KTH Royal Institute of Technology.

Byfors J (1980): Plain Concrete at Early Ages, Swedish Cement and Concrete Research Institute, FO 3:80, Stockholm.

Carlsson F, Plos M, Norlin B & Thelandersson S (2008): Säkerhetsprinciper för bärighetsanalys av broar med icke-linjära metoder. Report TVBK-3056, Lunds University, Sweden. (In Swedish.)

Castilho E (2003): Thermal analysis of concrete dams during construction: application to Alqueva’s Dam. Portugal: University of Lisbon, 2003.

Cervenka V, Jendele L & Cervenka J (2016): Atena Program documentation – Part 1 Theory. Cervenka Consulting, Prague.

Cervenka V & Gerstle K (1971): Inelastic Analysis of Reinforced Concrete Panels. Part I: Theory, IABSE, Vol 31, pp 32-45.

Cervenka V & Gerstle K (1972): Inelastic Analysis of Reinforced Concrete Panels. Part II: Experimental Verification, IABSE, Vol 32, pp 26-39.

Chen Y, Wang C, Li S, Wang R & He J (2001): Simulation analysis of thermal stress of RCC dams using 3-D finite element relocating mesh method. Advances in Engineering Software, 32 (2001), 677-682.

Chopra A. K (1968): Earthquake Behavior of Reservoir-Dam Systems. Journal of Engineering Mechanics ASCE 96 (EM4), 1475-1500.

Chopra A. K (2008): Earthquake analysis of arch dams: Factors to be considered. In: The 14th World Conference on Earthquake Engineering. 12-17 October, Beijing, China.

Clough R.W & Penzien J (1993): Dynamics of Structures, 2nd Edition, McGraw-Hill, Singapore.

Cook, R.D, Malkus, D.S, Plesha, M.E & Witt R.J (2002): Concepts and applicatins of finite element analysis. 4th edition, John Wiley & Sons, New York.

Cornelissen H, Hordijk D & Reinhardt H (1986): Experimental determination of crack softening characteristics of normal weight and lightweight concrete. Heron, Vol. 31, No. 2, Delft.

Cotoi T (2015): Dam engineering GRV BUTT. Available from [accessed 2016-02-18] http://www.scribd.com/doc/280436707/Dam-Engineering-GRV-BUTT

Crisfield M & Willis J (1989): The analysis of reinforced concrete panels using different concrete models. Journal of Engineering mechanics, ASCE Vol 115, No. 3, pp 578-597.

Dabre G (2000): State of practice in earthquake analysis of dams. Availiable from [accessed 2016-02-18] http://www.zlg.ethz.ch/downloads/publ/publ_bl15/darbre.pdf

Dahlblom, O & Lindemann J (2000): A program for simulation of temperature and stress in hardening concrete. Lund Technical University, Structural Mechanics.

Dameron R.A, Rashid Y.R & Hessheimer M.F (2003): Posttest analysis of a 1:4-scale prestressed concrete containment vessel model. IN: Transactions of the 17th Internationeal Conference on Structural mechanics in reactor technology (SMiRT 17), August 17-22, Prague.

Dassault (2014). Abaqus Analysis User's Manual. Dassault Systèmes Group.

de Borst R, Remmers J, Needleman A & Abellan M-A. (2004) Discrete vs smeared crack models for concrete fracture: bridging the gap. International journal for numerical and analytical methods in geomechancis, 2004, Vol. 28, pp 583 – 607.

Ekström T, Gustafsson P-J, Hallgren M, Hassanzadeh M, Malm R, Nilsson L-O & Thelandersson S (2016): Vägledning för granskning av avancerade dator-beräkningar avseende mekanik och transportprocesser i betong-konstruktioner för vattenkraft. Energiforskrapport 16:xx (in press), Energiforsk.

Elsayed A (2012): Experimental Study of the Shear Strength of Unfilled and Rough Rock Joints. MSc Thesis, Department of Civil and Architectural Engineering, KTH Royal Institute of Technology.

Eurocode 1 (2011): Actions on structures. Part 1-1: General actions – Densities, self-weight, imposed loads for buildings. SS EN 1991-1-1, SIS.

Eurocode 1-5 (2003): Actions on structures. Part 1-5: General actions – Thermal actions. SS EN 1991-1-1-5, SIS.

Eurocode 2 (2008): Design of concrete structures. Part 1-1: General rules and rules for buildings. SS EN 1992-1-1:2005, SIS.

Eurocode 7 (2010): Geotechnical design – Part 1: General rules. SS EN 1997-1:2005, SIS.

Eurocode 8 (2009): Design of structures for earthquake resistance. Part 1-1: General rules, seismic actions and rules for buildings. SS EN 1998-1-1:2004, SIS.

Eurocode 8-5 (2009): Design of structures for earthquake resistance. Part 5: Foundations, retaining structures and geotechnical aspects. SS EN 1998-1-5:2004, SIS.

Eriksson A (2002): Lecture notes from course 1C1117 Finite Element Methods, KTH Royal Institute of Technology.

Felippa C (2004): Introduction to Finite Element Methods. Course material University of Colorado, Boulder. Download from [accessed 2016-02-18]: http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/Home.html

FERC (1997): Engineering Guidelines for the Evaluation of Hydroelectric Projects- Chapter 10 Other dams.

FERC (1999): Engineering Guidelines for the Evaluation of Hydroelectric Projects- Chapter 11 Arch dams.

FERC (2002): Engineering Guidelines for the Evaluation of Hydroelectric Projects- Chapter 3. Gravity dams.

fib Model Code (2010): fib Model Code 2010, bulletins 55 and 56, International Federation for Structural Concrete, Lausanne, Switzerland, 2012.

Fishman Y. A (2007): Features of shear failure of brittle materials and concrete structures on rock foundations. International Journal of Rock Mechanics and Mining Sciences 45 (6), 976–992.

Fishman Y. A (2009) Stability of concrete retaining structures and their interface with rock foundations. International Journal of Rock Mechanics and Mining Sciences 46 (6), 957–966.

Fu C & Hafliðason B (2015): Progressive failure analyses of concrete buttress dams : Influence of crack propagation on the structural dam safety. TRITA-BKN-Examensarbete, 457. KTH Royal Institute of Technology.

FWHA (2011): Portland cement concrete pavements – Thermal coefficient of Portland cement concrete. Federal Highway Administration Research and Technology. Available from [accessed 2016-02-18]: http://www.fhwa.dot.gov/publications/research/infrastructure/pavements/pccp/thermal.cfm

Gasch T, Facciolo L, Eriksson D, Rydell C & Malm R (2013): Seismic analyses of nuclear facilities with interaction between structure and water – Comparison between methods to account for Fluid-Structure-Interaction (FSI). Elforsk report 13:79, Elforsk AB.

Gasch T, Malm R & Ansell A (2016): A coupled hygro-thermo-mechanical model for concrete subjected to variable environmental conditions. Submitted to International Journal of Solids and Structures.

Goldgruber M (2015): Nonlinear Seismic Modelling of Concrete Dams. PhD Thesis, Graz Technical University

Goldgruber M & Malm R (2014): Nonlinear seismic simulation of an arch dam using XFEM. In: Abaqus Users conference, Graz.

Gustafsson A, Johansson F, Rytters K & Stille H (2008): Betongdammars glidstabilitet – förslag på nya riktlinjer. Elforsk report 08:59.

Hellgren R (2014): Influence of Fluid Structure Interaction on a Concrete Dam during Seismic Excitation. MSc Thesis, Department of Civil and Architectural Engineering, KTH Royal Institute of Technology.

Hillerborg A, Modéer M, Petersson P (1976): Analysis of crack formation and growth in concrete by means of fracture mechanics and finite elements. Cement and Concrete Research, Vol 6, No. 6, pp. 773-782.

Holt E (2001): Early Age Autogenous Shrinkage of Concrete. PhD thesis, University of Washington, Seatte, 209 pp.

ICOLD (1986): Static analysis of embankment dams, Bulletin 53, International Commission of Large Dams (ICOLD).

ICOLD (1987): Finite elements methods in analysis and design of dams. ICOLD Bulletin 30.

ICOLD (1988): Dam design criteria – Philosophy of choice, Bulletin 61, International Commission of Large Dams (ICOLD).

ICOLD (1994): Computer software for dams – Validation, comments and proposals, Bulletin 94, International Commission of Large Dams (ICOLD).

ICOLD (2001): Computational procedure for dam engineering – Reliability and applicability, Bulletin 122, International Commission of Large Dams (ICOLD).

ICOLD (2010): Selecting seismic parameters for large dams – guidelines (revision of bulletin 72), Bulletin 148, International Commission of Large Dams (ICOLD).

ICOLD (2013): Guidelines for use of numerical models in dam engineering, Bulletin 155, International Commission of Large Dams (ICOLD).

ICOLD (2016): The physical properties of hardened conventional concrete in dams, Bulletin 145, International Commission of Large Dams (ICOLD).

ICOLD (201x): Non-linear modelling of concrete dams (in preparation), Bulletin XXX, International Commission of Large Dams (ICOLD).

Irwin G.R. (1958). Fracture. Encyclopedia of Physics (Handbuch der Physic)", Vol VI, Flügge (Ed.), Springer Verlag, Berlin, pp. 551-590.

Isander A, Nilsson C-O, Leblanc M & Malm R. (2013): Evaluation and Structural Rehabilitation of Storfinnforsen Concrete Buttress Dam (In French). In: CFBR Modernisation des barrages, Chambéry, December 4 – 5, 2013. Available from http://www.barrages-cfbr.eu/IMG/pdf/4.06.rehab_structure_barrage_storfinnforsen_suede.pdf

Ishikawa M (1991): Thermal Stress Analysis of a Concrete Dam. Computers & Structures, Vol 40 (2), pp. 347 -352.

James R.J & Dollar D.A (2003): Thermal Engineering for the Construction of Large Concrete Arch Dams, In: The 6th ASME-JSME Thermal Engineering Joint Conference, March 16-20

JCSS (2001): Probabilistic model code. Available from [accessed 2016-02-18] http://www.jcss.ethz.ch/

Jirasek M (2014): Modeling of localized inelastic deformation. Course material from Advanced course on modelling of localized inelastic deformation, Czech Technical University.

Johansson F, Ekström I, Rito Pi C, Malm R, Carlsson V (2015): FEM-analysis of a concrete dam in northern Sweden. In: ASDO (Association of state dam safety officials), New Orleans, September 2015.

Johansson F (2009): Shear Strength of Unfilled and Rough Rock Joints in Sliding Stability Analyses of Concrete Dams. Trita-JOB. PHD, 1013, KTH Royal Institute of Technology, Stockholm, Sweden.

Karihaloo B (2003): Failure of concrete. Comprehensive Structural Integrity, Vo. 2.10, 475 – 546.

Kolmar W (1986): Beschreibung der kraftuebertragung über rise in nichtlinearen finite-element-berechnungen von stahlbetontragwerken. (In German). PhD Thesis, Darmstadt University of Technology, Germany.

Kuo, J.S (1982): Fluid-Structure Interactions: Added mass Computation for Incompressible Fluid. UCB/EERC-82/09 Report, University of California, Berkely.

Kölfors J (1994): Temperatursprickor i grova betongkonstruktioner. Rapport 94:3B. Elforsk AB.

Leclerc M, Léger P & Tinawi, R (2003): Computer aided stability analysis of gravity dams: CADAM. Advances in Engineering Software, Vol 34 (7), pp 403 – 420.

Léger P & Seydou S (2009) Seasonal Thermal Displacements of Gravity Dams Located in Northern Regions. Journal of Performance of Constructed Facilities, ASCE, Vol. 23, No 3, pp. 166-174.

Linsbauer H & Bhattacharjee S (1999): Dam safety assessment due to uplift pressure action in a dam-foundation interface crack. In: Proceedings Fifth Benchmark Workshop on Numerical Analysis of Dams, USCOLD.

Lundqvist P & Nilsson L-O (2011): Evaluation of prestress losses in nuclear reactor containments, Nuclear Engineering and Design 241, p. 168-176, 2011.

Malla S & Wieland M (1999): Analysis of an arch-gravity dam with a horizontal crack. Computers & Structures, 72, pp 267 – 278.

Malm R, Eriksson D, Gasch T & Hassanzadeh M (2011): Probabilistic analyses of thermal induced cracking in a concrete buttress dam. In: XI Benchmark workshop on numerical analysis of dams, Valencia, October 20-21.

Malm R, Gasch T, Eriksson D & Hassanzadeh (2013): Evaluating stability failure modes due to cracks in a concrete buttress dams. In: 81st ICOLD Annual Meeting "Changing Times: Infrastructure Development to Infrastructure Management". 12 – 16 August 2013, Seattle, USA.

Malm R, Hassanzadeh M, Gasch T, Eriksson D, Nordström E. (2013b) Cracking in the concrete foundation for hydropower generators – Analyses of non-linear drying diffusion, thermal effects and mechanical loads. Energiforsk report 13:63, Energiforsk, Stockholm.

Malm R, Holmgren J (2008): Cracking in deep beams owing to shear loading – Part 2: Non-linear analysis. Magazine of Concrete Research, Vol. 6, No. 5, pp. 381-388.

Malm R (2006): Shear cracks in concrete structures subjected to in-plane stresses. Licentiate Thesis, Bulletin 88. Department of Civil and Architectural Engineering, KTH Royal Institute of Technology, Stockholm, Sweden.

Malm R (2012): Low-pH concrete plug for sealing the KBS-3V deposition tunnels. Report R-11-04, Swedish Nuclear Fuel and Waste Management Co (SKB), Stockholm.

Malm R & Ansell A (2011): Cracking of a Concrete Buttress Dam Due to Seasonal Temperature Variation. ACI Structural Journal 108 (1), pp 13-22.

Malm R (2015): Non-linear analyses of concrete beams with Abaqus. Course compendium for AF2102 Concrete Structures, advanced course. Department of Civil and Architectural Engineering, KTH Royal Institute of Technology, Stockholm.

Malm R (2009): Predicting shear type crack initiation and growth in concrete with non-linear finite element method, PhD Thesis, Bulletin 97. Department of Civil and Architectural Engineering, KTH Royal Institute of Technology, Stockholm, Sweden.

Malm R, Nordström E, Nilsson C-O, Tornberg R & Blomdahl J (2016a): Analysis of potential failure modes and instrumentation of a concrete dam. In: 84th ICOLD Annual Meeting "Appropriate Technology to Ensure Proper Development, Operation and Maintenance of Dams in Developing Countries", 15 – 20 May 2016, Johannesburg, South Africa

Malm R, Johansson F, Hellgren R & Rios Bayona F (2016b): Load capacity of grouted rock bolts (in preparation). Energiforsk report.

Mang H, Lackner R, Meschke G & Mosler J (2003): Computational modelling of concrete structures. Comprehensive Structural Integrity, Vol. 3.10, 536–601.

McKay M. & Lopez F (2014): Practical methodology for inclusion of uplift and pore pressures in analysis of concrete dams, IPENZ Proceedings of Technical Groups 39.

Messer C, Lopez F & Laxman M (2011): Finite Element Thermal Analysis of the Enlarged Cotter Dam. In: AnCold.

Mikola R.G & Sitar N (2013): Seismic Earth pressures on Retaining Structures in Cohesionless Soils. Report No. UCB GT 13-01, University of California, Berkeley.

NEA (2005): International standard problem No. 48 Containment capacity – Synthesis Report. Nuclear Energy Agency (NEA), NEA/CSNI/R(2005)5/VOL1.

Ngo D & Scordelis A.C (1967): Finite Element Analysis of Reinforced Concrete Beams, Journal of ACI, Vol. 64, No. 3, pp. 152-163.

Niu Y-Z, Tu C-L, Liang R.Y & Zhang S-W (1995): Modeling of Thermomechanical Damage of Early-Age Concrete. Journal of Structural Engineering, Vol 121 (4) pp. 717-726.

Nordström E, Malm R, Blomdahl J, Tornberg R & Nilsson C-O (2015): Optimization of Dam monitoring for long concrete buttress dams. In: 25th ICOLD Congress, Q99-R14, 13 – 20 June 2015, Stavanger, Norway

NW-IALAD (2006): Integrity Assessment of Large Concrete Dams. European Commission project G1RT-CT-2002-05076. The project webpage (www.nw-ialad.uibk.ac.at) is no longer functioning, project description is found at http://cordis.europa.eu/project/rcn/62813_en.html

Pacoste C (2001): Lecture notes from course 5C1860 FEM Modelling, KTH Royal Institute of Technology.

Pacoste C, Plos M & Johansson M (2012): Recommendations for finite element analysis for the design of reinforced concrete slabs. KTH Royal Institute of Technology, TRITA-BKN Report 144.

Rashid Y.R (1968): Ultimate Strength Analysis of Pre-stressed Concrete Pressure Vessels, Nuclear Engineering and Design, Vol 7, No. 4, pp 334 -344.

RIDAS (2011): Swedish Hydropower companies guidelines for dam safety, application guideline 7.3 Concrete dams (In Swedish). Svensk energi.

Rijkswaterstaat (2012): Guidelines nonlinear Finite Element Analysis of Concrete Structures, Scope: Girder Members. Document RTD 1016:2012, Ministry of Infrastructure and Environment, the Netherlands.

Rots J.G (1988): Computational Modeling of Concrete Fracture. PhD Thesis, TR diss 1663, Civil Engineering and Geosciences, TU Delft, Netherlands.

Rueda F, Camprubi N & Garcia G (2005): Thermal cracking Evaluation for La Breña II Dam during the Construction Process, In: Abaqus Users Conference.

Ruggeri G (2004): Sliding safety of existing gravity dams: final report. Working Group on Sliding Safety of Existing Gravity Dams. ICOLD European Club.

Rydell C (2014): Seismic high-frequency content loads on structures and components within nuclear facilities. Tech Lic. Bulletin 123. Department of Civil and Architectural Engineering, KTH Royal Institute of Technology, Stockholm, Sweden.

Saoma V.E & Ingraffea A.R (1981): Fracture Mechanics Analysis of Discrete Cracking. Proceedings of IABSE Colloquium In Advanced Mechanics of Reinforced Concrete, Delft, June 1981, pp. 393-416.

Shahriari S (2013): Fluid Structure Interaction – Arch Dam-Reservoir at Seismic Loading. In: ICOLD 12th Benchmark workshop on Numerical analysis of Dams, 2nd -4th October, Graz, Austria.

SKI (1992): Characterization of seismic ground motions for probabilistic safety analysis of nuclear facilities in Sweden, SKI Technical Report 92:3, National Nuclear Power Inspectorate (Statens Kärnkraftsinspektion, SKI), Stockholm, Sweden.

Spears R.E & Jensen S.R (2012): Approach for selection of Rayleigh damping parameters used for time history analysis. Journal of Pressure Vessel Technology 134.

Sustainable Bridges (2007): Guideline for load and resistance assessment of existing European railway bridges: advices on the use of advanced methods. Prepared by Sustainable Bridges – a project within EU FP6.

Svensen D (2016) Numerical analyses of concrete buttress dams to design dam monitoring, MSc Thesis, TRITA-BKN-Examensarbete 492, KTH Royal Institute of Technology.

Swiss Committee on Dams (2003): Methods of analysis for the prediction and the verification of dam behavior.

Tassios, T & Vintzēleou, E (1987): Concrete-to-concrete friction. Journal of Structural Engineering, Vol. 114, pp. 832–849.

USACE (1989): Retaining and Flood Walls. US Army Corps of Engineers. EM 1110-2502. US Army Corps of Engineers.

USACE (1992): Strength design for reinforced-concrete hydraulic structures, EM 1110-2-2104. US Army Corps of Engineers.

USACE (1994): Arch dam design, EM 1110-2-2201. US Army Corps of Engineers.

USACE (1995): Gravity dam design, EM 1110-2-2200. US Army Corps of Engineers.

USACE (1999): Response spectra and seismic analysis for concrete hydraulic structures, EM 1110-2-6050. US Army Corps of Engineers.

USACE (2003): Time-History Dynamic Analysis of Concrete Hydraulic Structures. EM 1110-6051. US Army Corps of Engineers.

USACE (2007): Earthquake design and evaluation of concrete hydraulic structures, EM 1110-2-6053, US Army Corps of Engineers.

USBR (2006) State-of-Practice for the Nonlinear Analysis of Concrete, US Bureau of Reclamation.

Westberg M (2010): Reliability-based assessment of concrete dam stability. PhD Thesis. Lund Institute of Technology, Lund University.

Westberg M & Johansson F (2015): Probabilistic Model Code for Concrete Dams - Draft Version. Energiforsk report, Stockholm.

Westergaard, H.M (1933): Water Pressure on Dams during Earthquakes. Transactions, ASCE, Vol. 98, 418-472.

Zangeneh Kamali A, Svedholm C & Johansson M (2013): Effects of restrained thrmal strains in transversal direction of concrete slab frame bridges. TRITA-BKN report 149, Department of Civil and Architectural Engineering, KTH Royal Institute of Technology.

GUIDELINE FOR FE ANALYSES OF CONCRETE DAMS

Den här rapporten beskriver finita elementanalyser och behandlar olika aspekter som beräkningsingenjörer ställs inför då de ska genomföra strukturmekaniska analyser av betongdammar. Syftet är att rapporten ska kunna användas av de som genomför strukturmekaniska finita elementberäkningar på betongdammar. Rapporten innehåller även rekommendationer om modelleringsaspekter och materialegenskaper. I ett kapitel finns dessutom praktiska modelleringstips.

Finita elementanalyser används bland annat för att utvärdera och dimensionera betongdammar vilket gör det möjligt att genomföra en mer noggrann och detaljerad analys jämfört med traditionella metoder.

Another step forward in Swedish energy research

Energiforsk – Swedish Energy Research Centre is a research and knowledge based organization that brings together large parts of Swedish research and development on energy. The goal is to increase the efficiency and implementation of scientific results to meet future challenges in the energy sector. We work in a number of research areas such as hydropower, energy gases and liquid automotive fuels, fuel based combined heat and power generation, and energy management in the forest industry. Our mission also includes the generation of knowledge about resource-efficient sourcing of energy in an overall perspective, via its transformation and transmission to its end-use. Read more: www.energiforsk.se

Energiforsk