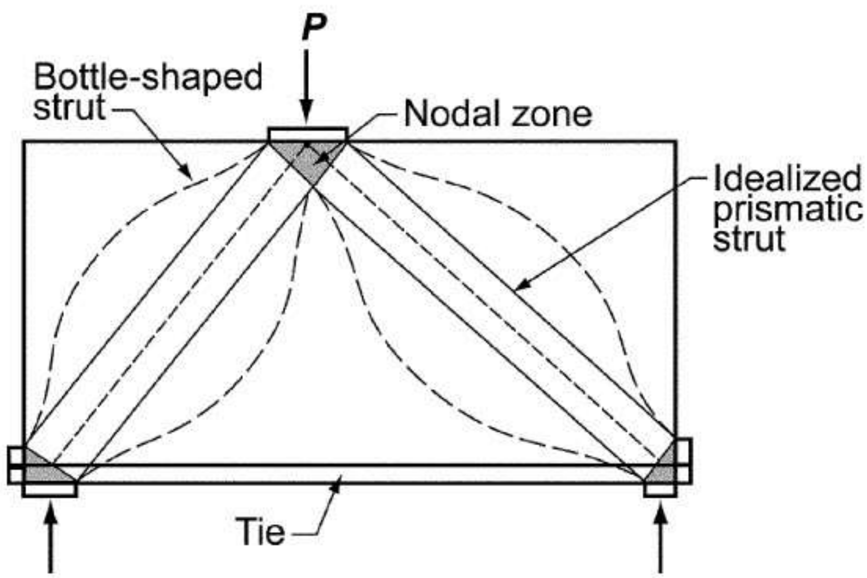
Strut-and-Tie-Modeling in Reinforced Concrete Structures

Basics and Applications



Adisorn Owatsiriwong, D.Eng.

ALPS Consultants, Bangkok, Thailand

Email: adisorn.alps@gmail.com

Revised: 05.02.2013

Preface

Strut-and-tie modeling technique is a simple and effective method which can be used as a quick tool for analysis of discontinuous region (D-region) in reinforced and prestressed concrete structures. It serves practicing engineers to grasp load transfer characteristics in order to provide good details of reinforcement and to determine load carrying capacity of the members in very effective way. Since the method is based on lower bound theorem of plasticity, it can be assured to deliver safe designed structure. Using of strut-and-tie modeling demands clear understanding of load path and good skill in visualization of stress field. Those understandings can be improved by studying several model examples and practicing with various design problems.

This course material aims to provide fundamental knowledge of strut-and-tie modeling starting from basic principles, basic of strut-and-tie modeling, review of ACI 318 Appendix A provision for strut-and-tie modeling. The last two chapters are related to software used for strut-and-tie modeling with worked examples. Many contents in this material are obtained from several excellent textbooks and technical papers that the author found to be useful and understandable. An interested reader may follow the list of references provided for further reading. Finally, the author hopes that this material will be useful not only for students but also practicing engineers to gain understanding in this topic and lead to improvement in their practices.

Contents

Chapter 1
Basic Principles

1.1 Discontinuous region and applications of STM

Discontinuous region (D-region) is the region where elementary beam theory (cf. Bernoulli's beam theory) ceases to be valid, i.e. plane section does not remain plane (due to nonlinear strain distribution), beam thickness does change (where normal stress along thickness direction cannot be neglected), and plane section does not remain normal to the neutral axis (shear strain may become large). D-regions are usually located where loads are applied or adjacent to support. They are also caused by load discontinuity due to openings or sudden change in model geometry like dapped end beam or stepped beam. Fig. 1.1 identifies common B- and D-regions in reinforced and prestressed concrete frame. To analyze complete stress and strain distribution in D-region, we need to apply continuum mechanics approach by considering state of equilibrium, compatibility and stress-strain relationship. Numerical methods like Finite Element Method (FEM), or Boundary Element Method (BEM) are commonly applied to solve such problems for arbitrary loads and model geometries.

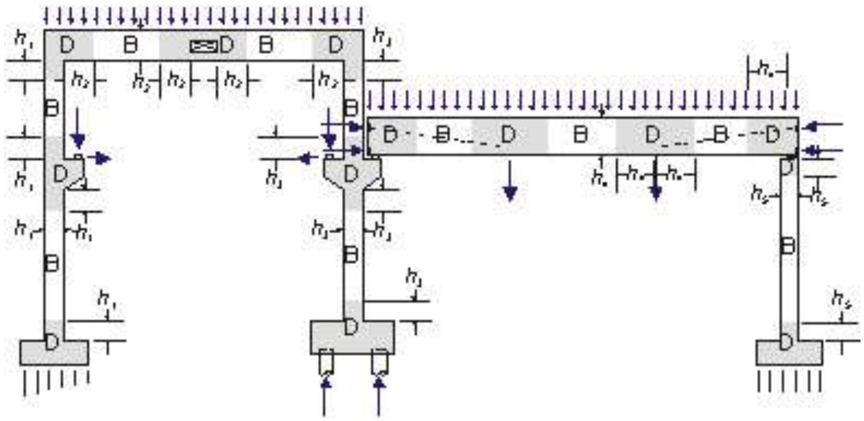


Figure 1.1 Identification of B- and D- regions in typical concrete frame

Should neither complete elastic stress (or strain) nor displacement of the D-region is necessary to be concerned as in typical reinforced concrete design problems, a simplified approach satisfying equilibrium without considering compatibility and constitutive relationship can be used. As proposed by Schlaich et al. (1987), Strut-and-Tie modeling (STM) can be used as an alternative approach for analysis and design of D-region. STM has been evolved and included in ACI 318 appendix A since ACI 318-02. A typical STM consists of struts, ties and nodal zones as shown in Fig. 1.2. Those three elements form in a truss-like structure in order to transfer loads to supports.

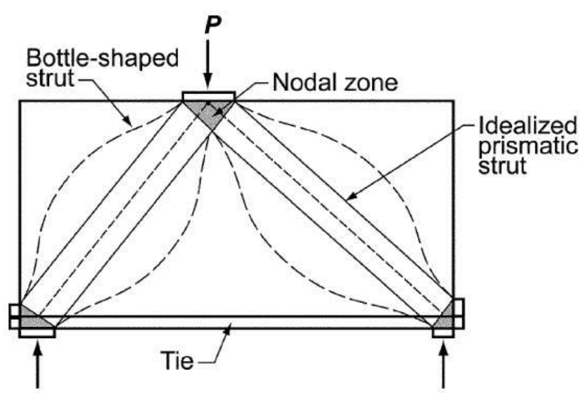


Figure 1.2 Typical strut-and-tie model (ACI 318M-08 Appendix A)

In STM methodology, factored axial loads in struts and ties are checked against their strength multiplied by reduction factor as written in the following expression:

$$\sum_{i=1}^{N} \psi_i F_i \leq \phi R_n \qquad (1.1)$$

where F_i = service load; $R_n =$ nominal strength; $\psi =$ corresponding load factors; $\phi =$ reduction factors

1.2 Lower bound theorem of theory of plasticity

STM relies on force equilibrium at nodes. The equilibrium equations applied at all nodes can be written by

$$\sum F_i = 0 \qquad \text{i = 1..ndim} \qquad \text{(at all nodes)} \qquad (1.2)$$

It should be noted that no compatibility requirement is required in STM. Furthermore, the influence of stiffness must be treated independently in statically indeterminate structure. Neither deformation (strain) nor stress distribution can be evaluated by STM.

The lower bound theorem of plasticity $[Q_S] \leq [Q_R]$ states that

"*A load system* $[Q_S]$*, based on a statically admissible field which nowhere violates the yield condition is a lower bound to the collapse load* $[Q_R]$" (Muttoni, Schwartz and Thuerlimann, 1997).

This implies that the solution obtained from the lower bound theorem (static method) usually lies on safe side or in conservative sense. The static method does not consider kinematic condition formed by plastic hinge mechanism. By neglecting plastic mechanism to be formed, STM will always give the collapse load below the true collapse load estimated by plastic analysis and hence safe design can be achieved. It should be noted that since different STMs also yield different results, the one that give *highest* factor of safety against collapse load shall be adopted as it is still less than the true collapse load. On the other hands, for a given set of loads, the model having minimum strain energy or giving minimum rebar amount shall be the best utilized model. Fig. 1.3 summarizes the solution by plastic theory. The true collapse load is the load where static solution (lower bound) meets kinematic solution (upper bound).

condition	static solution	complete solution	kinematic solution
equilibrium	ok	ok	ok
yield condition	ok	ok	?
mechanism	?	ok	ok
result	lower bound $[Q_S] \leq [Q_R]$	collapse load $[Q_R]$	upper bound $[Q_K] \geq [Q_R]$
method	static method	———	mechanism method

Figure 1.3 Overview of plastic solution methods (Muttoni et al., 1997)

1.3 Topology optimization and applications to STM

Topology optimization is a class of optimization techniques (size, shape, and topology) to reveal the most efficient material usage for a given set of loads and boundary conditions (Fig. 1.4). Topology optimization plays an important role in finding optimum STM and has become a root of several techniques use for optimization of STM. Among other techniques, Evolutionary Structural Optimization technique (ESO) (Xie & Steven, 1992; Xie and Steven, 1997; Huang & Xie, 2010) have been widely used and applied to well-posed 2D and 3D problems.

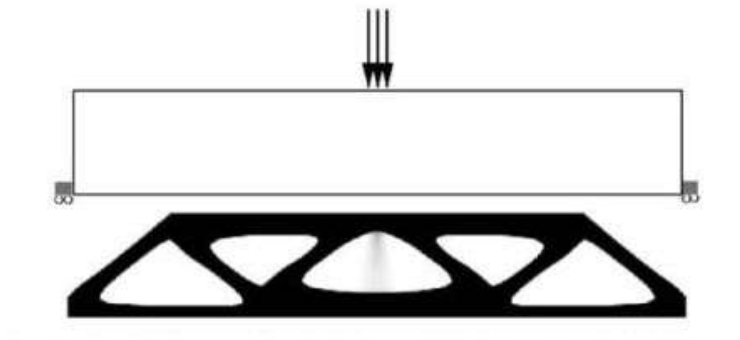


Figure 1.4 Topology optimization of a 2D structure (Cristensen and Klarbring, 2008)

1.3.1 Evolutionary Structural Optimization Technique

Evolutionary structural optimization (ESO) technique (Xie & Steven, 1992; Xie and Steven, 1997; Huang & Xie, 2010) is based on the simple concept of gradually removing inefficient material from a structure. Through this process, the resulting structure will evolve towards its optimal shape and topology. Many criteria can be used to reject part of material that deems inefficient such as level of stress, strain energy, and so on. One simple method to determine element removal is to consider the level of element stress, for example, the von Mises stress σ_{vm}^{e} of the element with the maximum von Mises stress of the whole structure $\sigma_{vm}^{\max}$. After each finite element analysis, elements which satisfy the following condition are deleted from the model.

$$\frac{\sigma_{vm}^{e}}{\sigma_{vm}^{\max}} < RR_i \tag{1.3}$$

where RR_i is the current rejection ratio (RR). Several cycles of finite element analysis are to be performed until the topology is unchanged under the constant RR, which means that no more element can be deleted using the current RR. By increasing RR by evolutionary ratio (ER) as

$$RR_{i+1} = RR_i + ER \tag{1.4}$$

After performing several cycles of analysis, more elements can be removed and the more efficient topology can be obtained.

Fig 1.5 shows the process of ESO in obtaining optimal topology of a deep beam subjected to a concentrated load.

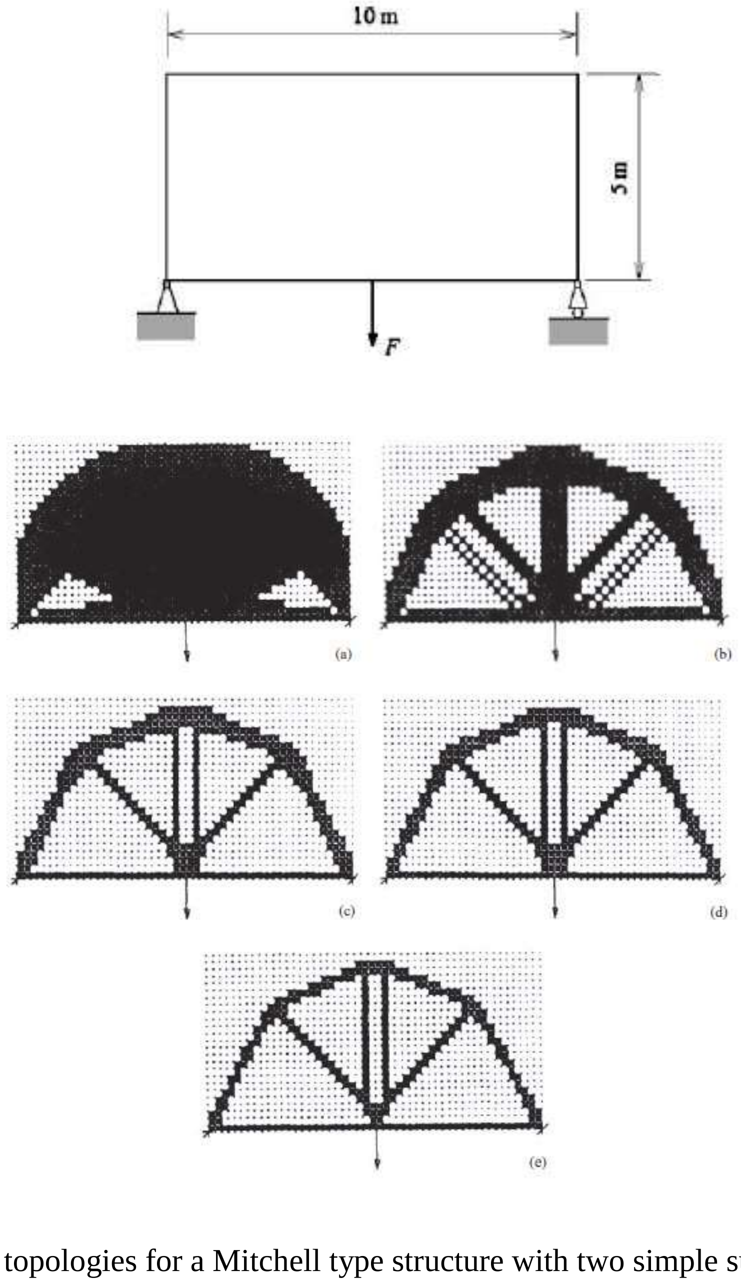


Figure 1.5 ESO topologies for a Mitchell type structure with two simple supports (Xie and Steven, 1993) (a) RR = 5%; (b) RR = 10%; (c) RR = 15%; (d) RR = 20%; (e) RR = 25%

1.3.2 Sensitivity Number and Evolutionary Procedure

Considering stress level in removing elements that subject to low stress is not suitable to the structure with limited plastic deformation like reinforced concrete. Sensitivity of the element to structural stiffness can be good criterion in removing the elements that contribute less to the overall stiffness of the structure.

Consider the mean compliance written in the form

$$C = \frac{1}{2}\mathbf{f}^T\mathbf{u} \tag{1.5}$$

where $\mathbf{f}$ is nodal force vector; $\mathbf{u}$ is displacement vector. In finite element analysis, static equilibrium of the structure is expressed as

$$\mathbf{Ku} = \mathbf{f} \tag{1.6}$$

where $\mathbf{K}$ is the global stiffness matrix.

Varying Eq.(1.6) and assume that external nodal force does not change,

$$(\Delta\mathbf{K})\mathbf{u} + \mathbf{K}(\Delta\mathbf{u}) = \Delta\mathbf{f} = 0 \tag{1.7}$$

$$\Delta\mathbf{u} = -\mathbf{K}^{-1}\Delta\mathbf{Ku} \tag{1.8}$$

It is simple to prove that change of global stiffness is equal to negative of the stiffness matrix of the elements that have been removed, so that

$$\Delta\mathbf{K} = -\mathbf{K}_i \tag{1.9}$$

Using Eqs. (1.5), (1.8), and (1.9), the change in the mean compliance can be determined from

$$\Delta C = \frac{1}{2}\mathbf{f}^T \Delta\mathbf{u} = \frac{1}{2}\mathbf{f}^T \mathbf{K}^{-1}\mathbf{K}_i \mathbf{u} = \frac{1}{2}\mathbf{u}_\mathbf{i}^\mathbf{T}\mathbf{K}_\mathbf{i}\mathbf{u}_i \tag{1.10}$$ *

*Since the component of $\mathbf{u}$ other than $\mathbf{u}_i$ does not contribute to strain energy to element i th, we can write $\mathbf{u} = \mathbf{u}_i$ for clarity.

where $\mathbf{u}_i$ is nodal displacement vector of element ith. Thus the sensitivity number of element th can be defined as

$$\alpha_i^e = \frac{1}{2}\mathbf{u}_\mathbf{i}^\mathbf{T}\mathbf{K}_\mathbf{i}\mathbf{u}_i \tag{1.11}$$

It can be noted that increase in the mean compliance as a result of the removal of the ith element is equal to its elemental strain energy. To minimize the mean compliance (which is equivalent to maximizing the stiffness) through the removal of elements, it is evident that the most effective way is to eliminate the elements which have the lowest value of α_i^e so that the increase in C will be minimal. The elemental removal ratio (ERR) as defined by the number of elements to be removed to the total number of elements can be used to update the initial or current analysis model.

The evolutionary procedure for stiffness optimization (maximize stiffness) is given as follows:

Step 1: Discretize the structure using a fine mesh of finite elements;

Step 2: Carry out finite element analysis;

Step 3: Calculate the sensitivity number for each element using (1.11);

Step 4: Remove a number of elements with the lowest sensitivity numbers according to a predefined element removal ratio ERR;

Step 5: Repeat steps 2 to 4 until the mean compliance (or the maximum displacement, etc.) of the resulting structure reaches a prescribed limit.

1.3.3 Evolutionary Structural Optimization with Minimum Weight

The previously described evolutionary structural optimization techniques do require the analyst to manually specify the portion of elements to be removed. Based on the sensitivity number described above Liang et al. (1999) proposed the topology optimization problem of a continuum structure which can be stated as follows:

$$\textit{minimize} \qquad W = \sum_{e=1}^{n} w_e\left(t_e\right) \tag{1.12}$$

$$\text{subject to} \qquad \left|u_j\right| - u_j^* \le 0 \qquad j = 1,..,m \tag{1.13}$$

where W is the total weight of the structure, w_e is the weight of the e^{th} element, t_e is the thickness of the e^{th} element that is treated as the design variables, $\left|u_j\right|$ is the magnitude of the jth displacement component, u_j^* is the prescribed limit of jth displacement, m is the number of displacement constraints and n is the total number of elements within structure.

To determine the sensitivity of the element to change in constrained displacement Δu_j, a unit load is applied to the jth displacement. Sensitivity number of the element subjected to unit load can be computed from

$$\Delta\mathbf{u}_j = \mathbf{u}_{ej}{}^T\mathbf{k}_e\mathbf{u}_e \tag{1.14}$$

where $\mathbf{u}_{ej}$ is the nodal displacement vector of the *e*th element for the unit load and $\mathbf{u}_e$ is the displacement vector of the *e*th element under the applied loads. Eq. (1.14) indicates the change in constrained displacement due to element removal, and can be used as a measure of the element efficiency.

The element with low sensitivity number will make the whole structure subject to small change when being removed. In other word, the elements that have the lowest sensitivity number do have little contribution to the stiffness of the structure and can be removed from the structure to achieve a more efficient design. Since concrete permits only limited plastic deformation, the strut-and–tie model with maximum stiffness or minimum deflections is the best while its weight is the minimum (Schlaich et al. 1987). Based on this fact, the ESO method for plane stress continuum structures subject to displacement constraints is appropriate for the topology optimization of strut-and-tie in non flexural reinforced concrete members. To minimize weight of the structure, performance index (PI) is used to measure the material efficiency in resisting deflection and failure of a plane stress structure. It can be defined as

$$PI = \frac{|u_{0j}|W_0}{|u_{ij}|W_i} \tag{1.15}$$

where W_0 is the initial weight of the structure; W_i is the weight of the structure at iteration *i*th; $|u_{0j}|$ is the magnitude of the *j*th nodal displacement in the initial design under the applied loads, and $|u_{ij}|$ is the jth nodal displacement in the current design at the ith iteration under applied loads. If the density is constant, performance index can be written by

$$PI = \frac{|u_{0j}|V_0}{|u_{ij}|V_i} \tag{1.16}$$

in which V_0 is the volume of the initial design domain and V_i is the volume of the current design at the *i*th iteration. Performance index varies reversely to the volume and nodal displacement of the current design. For a typical solution, PI will increase when the iteration bring the structure to be lighter. For a certain cycles of iteration when some effective elements are removed, the displacement of structure become larger and performance index becomes lower. The iteration shall be terminated when performance index become less than 1.0.

The optimization procedure is given as follows:

Step 1: Model the concrete member with a fine mesh of finite elements;

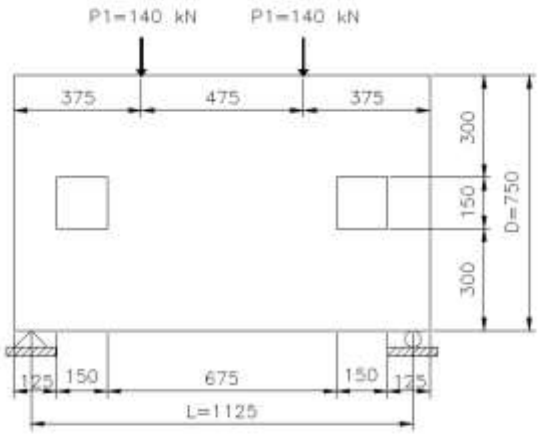
Step 2: Analyze the structure for the applied loads and unit loads;

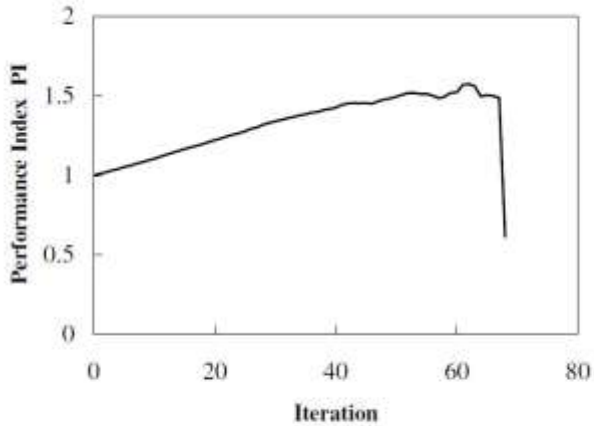
Step 3: Calculate the performance index using Eq. (1.6)

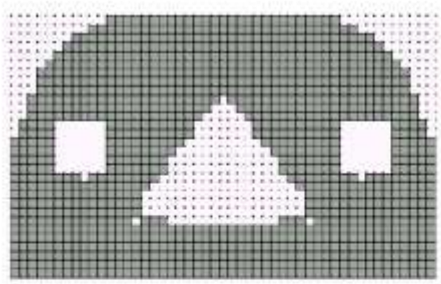
Step 4: Calculate the sensitivity number for each element using Eq. (1.14) or (1.15)

Step 5: Remove elements with the lowest sensitivity numbers;

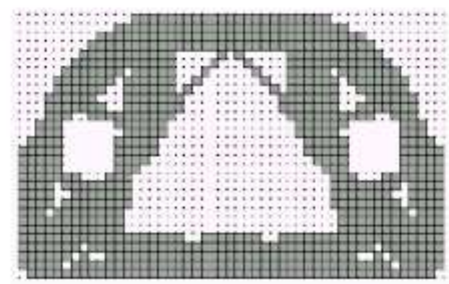
Step 6: Repeat step 2 to 5 until the performance index is less than unity.



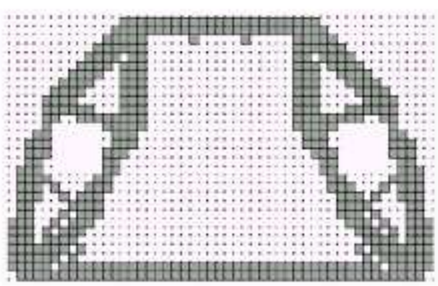




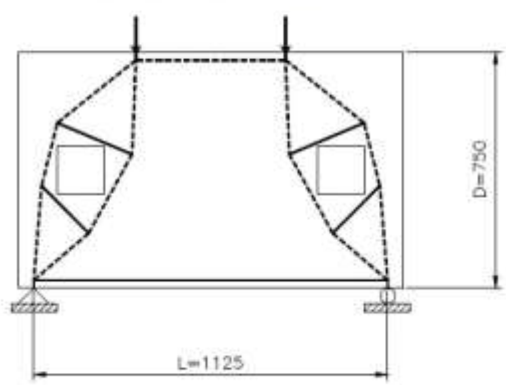
(a) Topology at iteration 20



(b) Topology at iteration 40



(c) Optimal topology



(d) Optimal strut-and-tie model

Figure 1.7 Optimization of strut-and-tie model in RC deep beam with web opening (Liang et al., 1999)

1.4 STM vs. FEM

Finite Element Method (FEM) is a powerful numerical method for solving partial differential equations. Based on the principle of virtual work or the principle of minimum total potential energy, the equilibrium equation can be formulated where displacement, stress, strain or all of them can be used as independent variables. The FE solution must satisfy the following conditions (Wilson, 2010):

- Force equilibrium (at node),
- Kinematic (strain-displacement) relationship, and
- Constitutive equations (at Gauss points)

Formulating a FE system requires four main steps (Bathe, 1996):

1. The structure or continuum is idealized as an assemblage of discrete elements connected at nodes pertaining to the elements.

2. The externally applied forces (body forces, surface traction initial stresses, concentrated loads) are lumped to these nodes using the virtual work principle to obtain equivalent externally applied nodal point forces.

3. The equivalent externally applied nodal point forces (calculated in 2)are equilibrated by the element nodal point forces that are equivalent (in the virtual work sense) to the element internal stresses; i.e., we have

$$\sum_m \mathbf{F}^{(m)} = \mathbf{R}$$

4. Compatibility and the stress-strain material relationship are satisfied exactly, but instead of equilibrium on the differential level, only global equilibrium for the complete structure, at the nodes, and of each element m under its nodal point force $\mathbf{F}^{(m)}$ *is satisfied.*

Generally, the FE system equation is sparse and symmetric where most non-zero values are located within the diagonal band of the system matrix ($\mathbf{K}$). Specific solvers employing those special characteristics like active column solver (Wilson, Bathe) can be used efficiently to solve such system. The nodal solution can be obtained by solving a system of linear algebraic equation. For displacement based FEM, nodal displacements are solved. Strain components at integration points can be computed from the strain-displacement relationship. Stress components at integration points are computed from constitutive relationship.

For assuring monotonic convergence of the FE solution, the finite elements used must also satisfy the following conditions:

- Completeness
- Compatibility

Completeness stands for the capability of the element to represent constant strain condition including zero strain as in case of rigid body motion. The element that is incapable to represent constant strain will never reach to the exact solution. Compatibility of the element inter-connection is not stringent as long as the elements can pass patch test. Incompatible (non-conforming) elements, if properly designed, can overcome locking problems as in case of shear locking and volumetric locking.

For displacement-based FEM, the solution of nodal displacement always brings the model to satisfy force equilibrium at nodes and at elements as shown in Fig. 1.7.

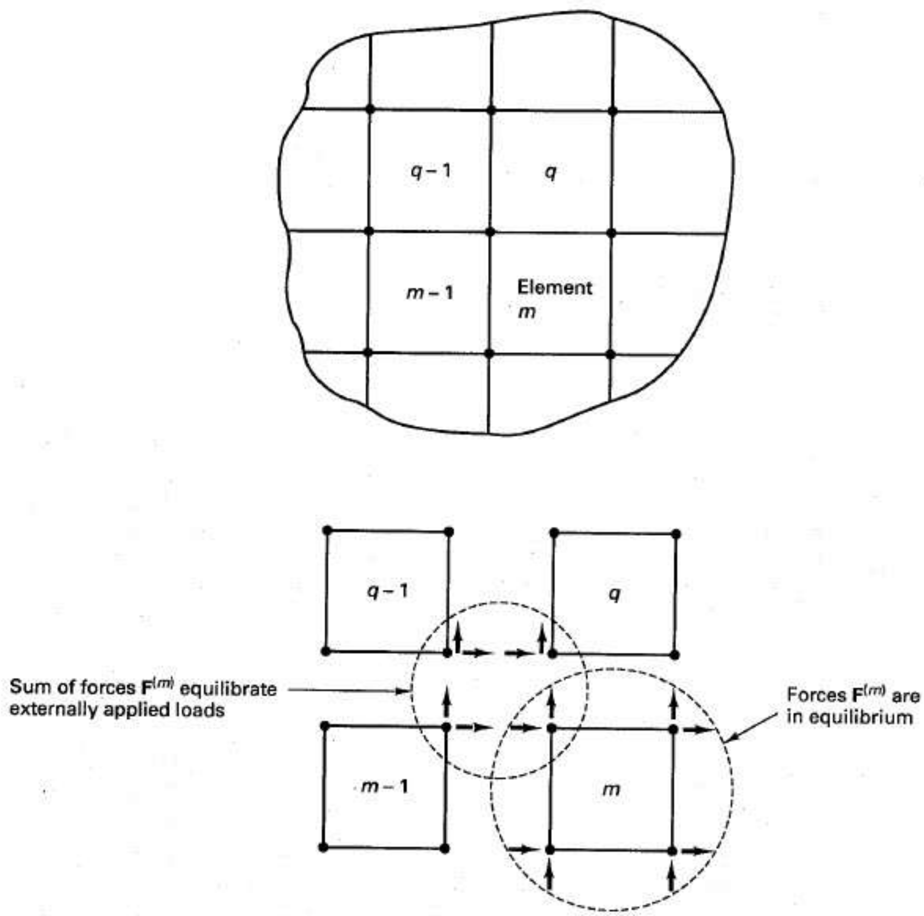


Figure 1.7 Nodal point and element equilibrium in a finite element analysis (Bathe, 1996)

However, continuous of stress or traction along inter-element boundary are not satisfied, except at a specific state of constant strain (stress). Hence, the element is known to satisfy equilibrium in 'global' sense. The accuracy of FE solution can be judged by considering smoothness of stress along element boundary. A useful concept of pressure band visualization (Bathe, 1996) can be used to evaluate stress discontinuities and accuracy of FE solutions as shown in Fig. 1.8.

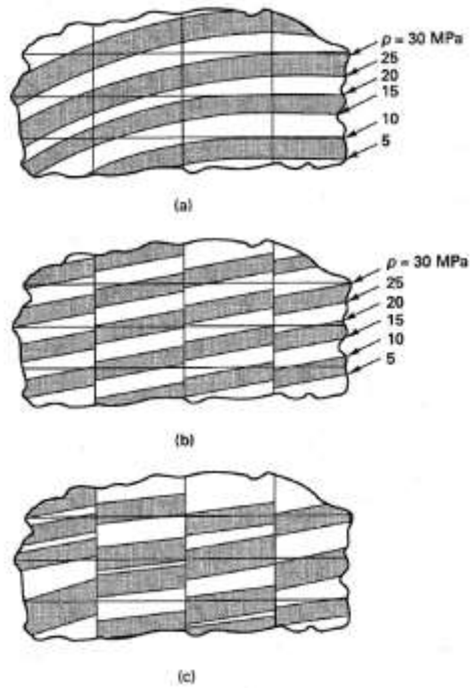


Fig. 1.8 Pressure band for estimating stress discontinuities, with of bands = 5 Mpa

(a) Smooth pressure band (b) visible discontinuities (c) significant discontinuities (Bathe, 1996)

To simplify the problem, STM employs only equilibrium condition at nodes without satisfying other two stringent conditions. It is therefore only applied to statically determinate system without any specific material characterization. The solution of STM then only contains element and nodal forces (reactions). It does not give any information about nodal displacement and stress (strain). However, as we use STM to model the equilibrium condition at limit state, elastic stress (strain) and deformation are not so relevant. Since both STM and (elastic) FEM employ the equilibrium equation without considering material yielding, they are expected to give the limit load less than theoretical collapse load as against to plastic mechanism analysis.

The solution of FE depends strongly to mesh quality (performance, aspect ratio, etc.) and mesh density. When the finer mesh is employed, the structure becomes less stiff and should lead to a better solution. The coarse mesh leads to smaller displacement and smaller stresses. Nonlinear finite element for plastic analysis is nothing related to the bound theorem of plasticity. The method fully employs nonlinear stress-strain relationship where plastic region can be determined by integrating stress over the domain as so-called *stress return algorithm*. The solution is therefore unique, at least for monotonic stress-strain increment. For elastic analysis, since equilibrium is applied, it is possible to use elastic FE solution to figure out an optimal ST model. Kanok-Nukulchai & Anwar (1996) employ FEM to come across optimal ST models of pilecaps with arbitrary pile arrangement. With aids of visualization technique, Dammika & Anwar (2010) demonstrate general approach to extract 3D ST models from FE solid element mesh for several complex structures such as pilecaps and bridge piers as shown in Fig. 1.9.

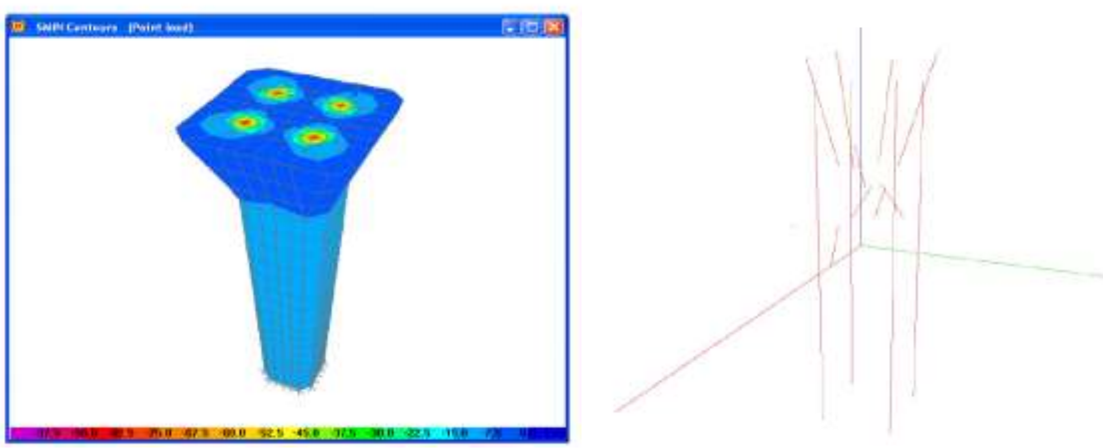


Figure 1.9 Principal Compressive Stress Contours of Pier Head under Point Loading & Extracted Strut and Tie Layout (Dammika & Anwar, 2010)

Chapter 2

Basic Strut-and-Tie Modeling

2.1 Struts

Struts are compression elements used to transfer loads from nodes to another nodes. Struts may be reinforced as columns or as typical walls. The influence of passing rebar must be considered in specifying compressive strength of struts. According to ACI 318-08 Appendix A, struts can be assigned as prismatic or bottle-shaped as shown in Fig. 2.1. Idealization of bottle-shaped strut is shown in Fig. 2.1 (b). The neck struts are assumed to incline at 2:1 (longitudinal: transverse).

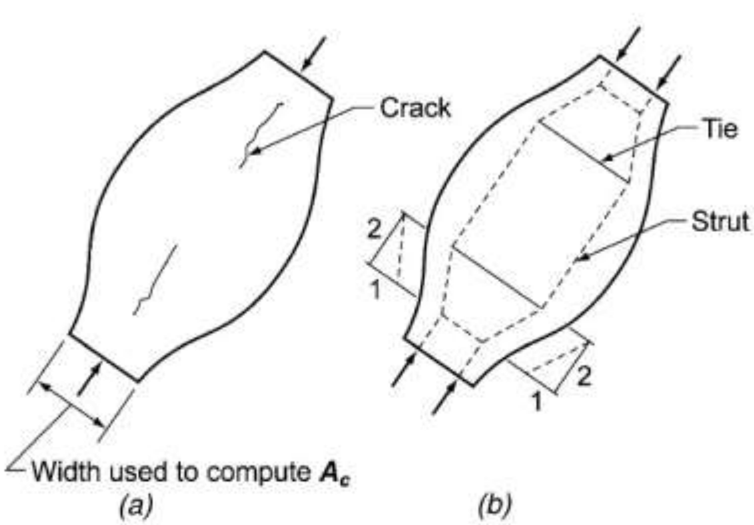


Figure 2.1 Bottle-shaped strut (a) Cracking of a bottle-shaped strut (b) strut-and-tie model of a bottle-shaped strut (ACI 318-08 App. A)

Bottle-shaped strut is usually located in the region where lateral spreading of compression field is possible; for example, at beam web. In practice, there is a rare need to model the idealized bottle-shaped strut as shown in Fig. 2.1, especially when quick hand calculation is performed. ACI 318 allows the bottle-shaped to strut to be modeled as prismatic or taper struts.

2.2 Ties

Ties are tension elements used to transfer tensile force to nodes. Ties can be prestressed or non-prestressing rebar surrounded by a portion of concrete that is concentric with the axis of ties. In design of tie element, care must be taken on providing sufficient anchorage length, so that full tensile strength of ties can be developed. For design purpose, it is assume that concrete within the ties does not carry any tensile force. The dimension of surrounding concrete is one factor to determine nodal zone dimension.

2.3 Nodal zones

Nodal zones are finite region that struts and ties are coincided. Nodal zones may be treated as a single region or may be subdivided into two smaller zones to equilibrate forces as shown in Fig, 2.2.

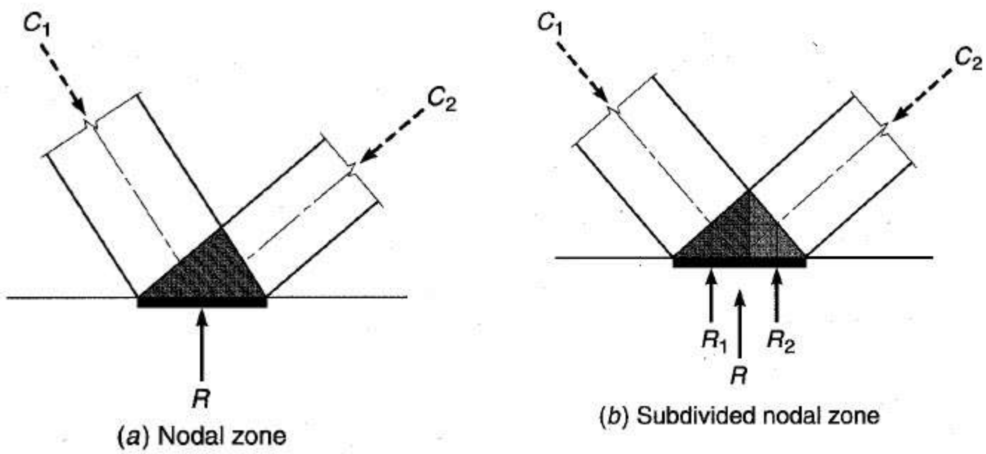


Figure 2.2 Subdivision of nodal zone

Depending on the direction of strut and ties joining at node, nodal zone is classified in to CCC, CCT, CTT and TTT nodes as shown in Fig. 2.3. Whatever nodal zones are defined, nodes are assumed to be in compression as they are under hydrostatic pressure.

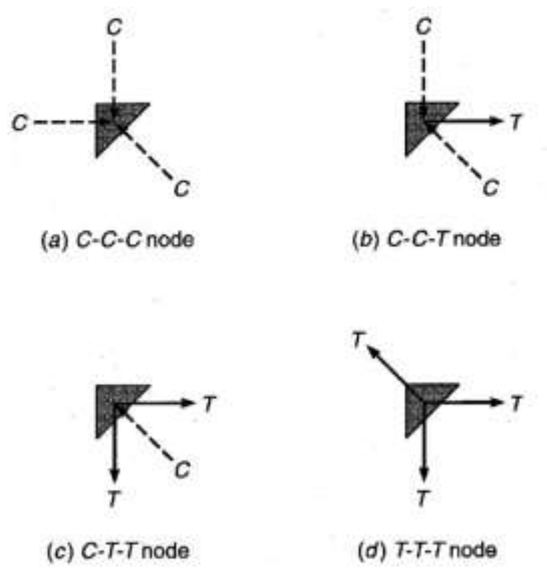


Figure 2.3 Classification of nodes

The hydrostatic pressure concept can be used to compute the dimensions of nodal zone. It can be assumed that the pressures at nodal faces are equal where tie is assumed to be yield. Consider CCC and CCT nodes in Fig. 2.4, the relationship between strut forces and node dimensions can be written by

$$\frac{C_1}{a_1} = \frac{C_2}{a_2} = \frac{C_3}{a_3} \tag{2.1}$$

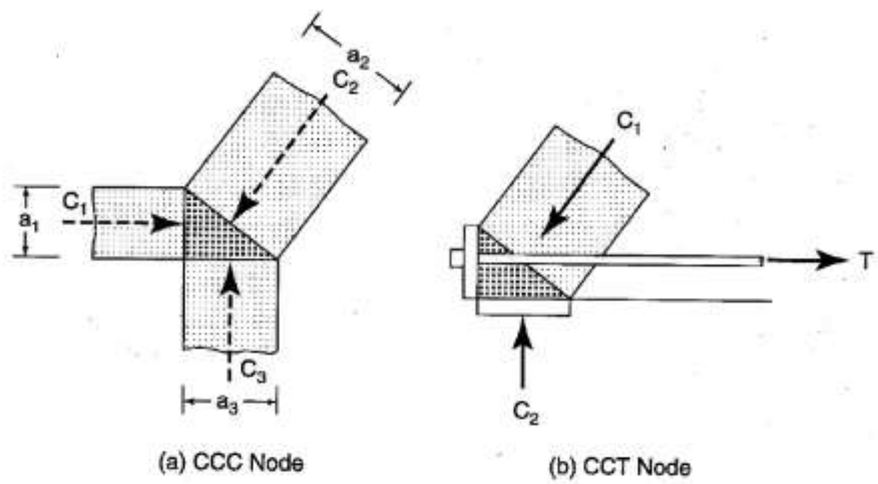


Figure 2.4 Nodal zone dimensions for a CCC and CCT node

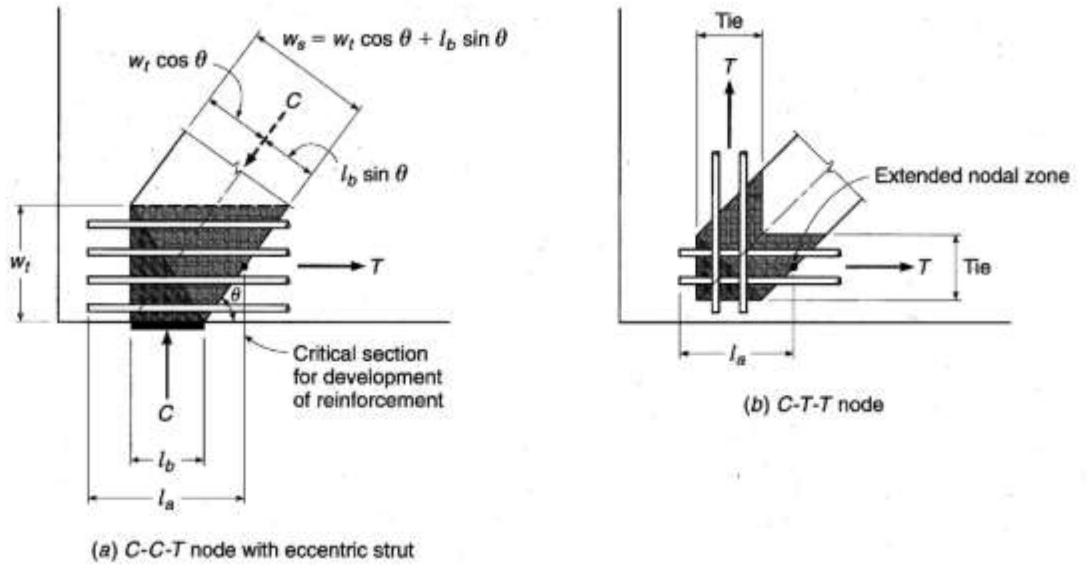


Figure 2.5 Extended nodal zone

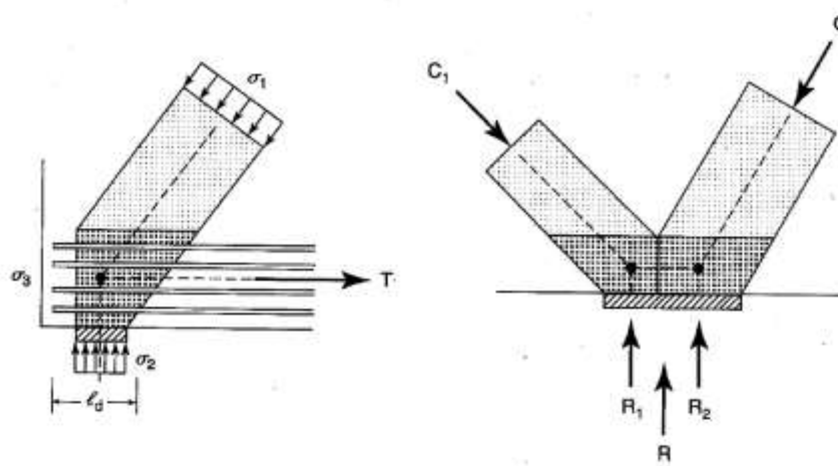


Figure 2.6 Nodal zones within the intersections of members

The use of hydrostatic elements sometimes does not represent the situation where nodes are confined by two boundary prescribed dimension as shown in Fig. 2.5 (a). According to Fig. 2.6, the stress σ_3 is determined from $\sigma_3 = T / A_3$ where A_3 is the area of vertical face. In contrast to hydrostatic concept, the value of σ_1, σ_2 and σ_3 can be different provided that

1. The resultants of the three forces coincide.

2. The stresses are within the limits as given by ACI 318 A.3.2, and A.5.2

3.The stress is constant on any one face.

Although, non-hydrostatic approach is not a standard method stipulated by ACI Code, it has been widely used by many researchers.

2.4 STM methodology

The following strut-and-tie modeling procedure proposed by MacGregor (1996) can be summarized as follows:

a. Define and isolate the D-regions.

b. Compute the internal stress on the boundaries of the element.

c. Subdivide the boundary and compute the force resultants on each sub-length.

d. Draw a truss to transmit forces from boundary to boundary of the D-region.

e. Check the stresses in the individual members of the truss.

2.5 Fundamental problems

Utilizing STM requires the designer to manually identify load path and nodal zone dimension, for which, in particular cases, they are not easily defined. However, for many common structures, STM models are available and well validated from experimental results. This section presents some fundamental STMs which can be used as guidelines for constructing the more complicated models. It should be mentioned that the best STM is not necessary the model with minimum amount of steel as theoretically stated, but it is the model that behaves right with compliance to good construction detailing and practice.

2.5.1 Transfer girder

Transfer girders are moderately deep beam with $3 < L/d \leq 5$. They are generally used to transfer load from the upper building stories down to the main columns below in order to enlarge clear space for lobby or parking area (Fig, 2.7). As shown in Fig. 2.8, diagonal strut can be idealized as prismatic or bottle-shaped strut. Although, bottle-shaped strut modeling is well replicated to experimental results as demonstrated in Fig. 2.9 (b), its reinforcement detail is quite complicated and found to be cumbersome with hand calculation. With aid of the computer software, analysis using bottle-shaped strut does not make any different effort from that of prismatic strut. Bottle-shaped strut shall be used when spreading of compression field is possible as in case of diagonal strut in deep beam as shown in Fig. 2.8. Code section A.3.3 requires that a minimum percentage of reinforcement be distributed across bottle-shaped struts to control cracking along the axis of the strut. This reinforcement can be provided either in an orthogonal mesh or as vertical-only or horizontal only reinforcement (Wight & Parra-Montesinos, 2003).

Prismatic strut, as simplified version of bottle-shaped strut, shall be used where spreading of compression field is not possible as in case of horizontal strut in compression face of deep girder.

Minimum reinforcement as stated by ACI A3.3 is necessary for crack control due to lateral spreading of compressive stress field.

Figure 2.7 Three-span deep beam, Brunwich building, Chicago. (JG MacGregor, 1997)

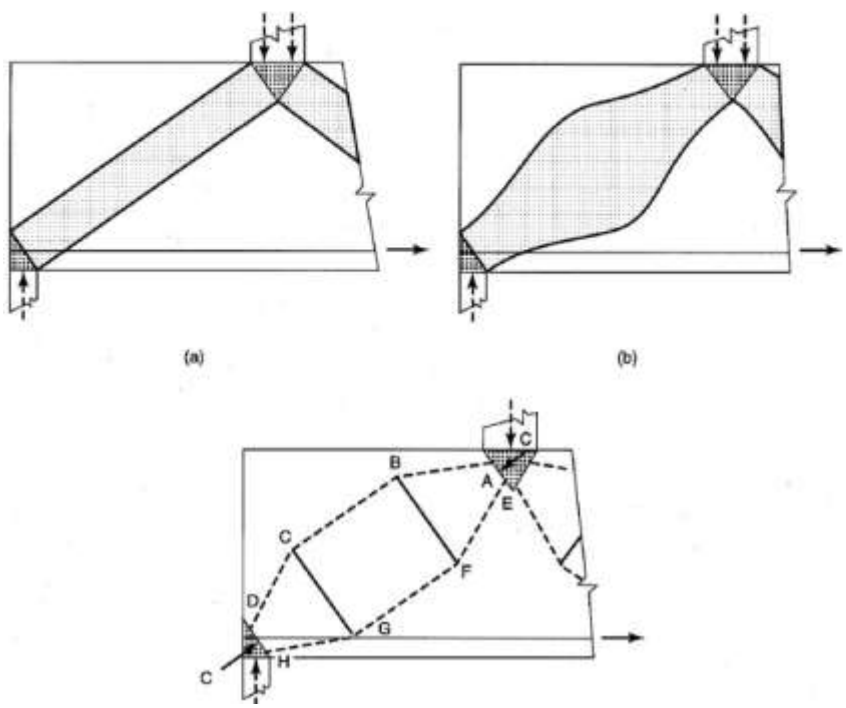
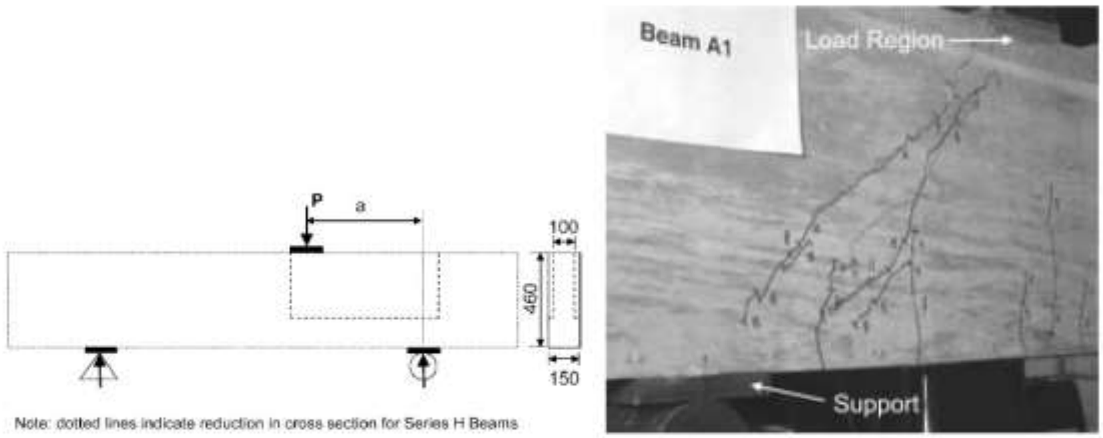


Figure 2.8 Strut-and-tie model of a deep girder to transfer load from single column

(MacGregor, 1997)

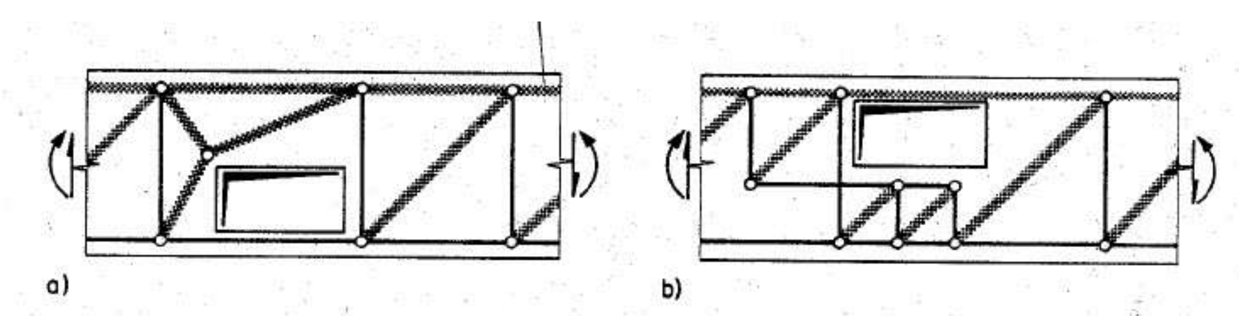


(a) (b)

Figure 2.9 (a) Experiment speciment (b) failure of deep beam indicating spreading of crack within compression strut region (Quintero-Febres et al., 2006)

2.5.2 Beam with openings

Openings in beam are sometimes required for conduits or pipes routing. Strut-and-tie modeling can be used to evaluate the edge reinforcement around opening and to investigate the adequate thickness of concrete close to the openings to prevent cracking by large compressive field. Fig. 2.10 shows strut-and-tie models for beam with web openings.



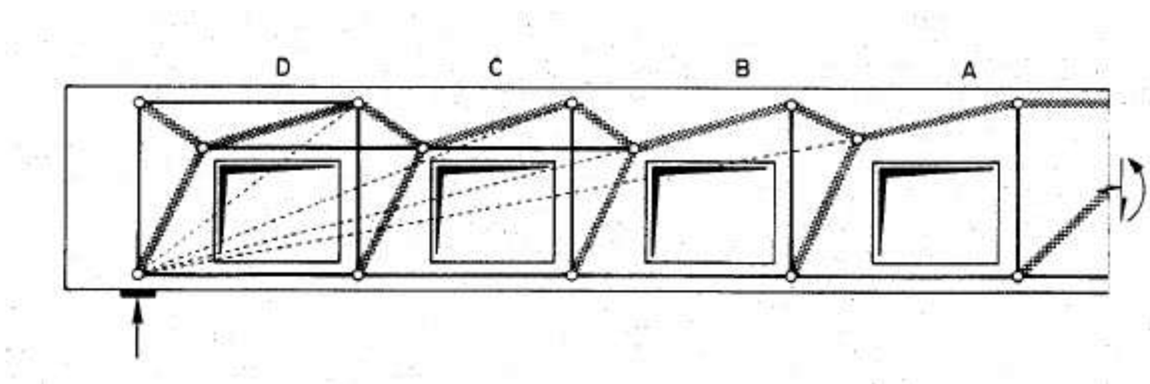


Figure 2.10 Truss model for beam with openings (Muttoni et al., 1997)

2.5.3 Bracket or corbel

Bracket or corbels are reinforced or prestressed concrete element to support vertical and horizontal load. They are typically projected from column face as shown in Fig. 2.11. Fig. 2.12 presents stress field for STM identification. It should be mentioned that ties must be properly anchored to the boundary face of the corbel as required by code of practice. They can be hooked with horizontal bar or welded to support plate. The latter detail is more preferred to prevent corner or face spalling as might occur as shown in Fig. 2.13 (b) or (c).

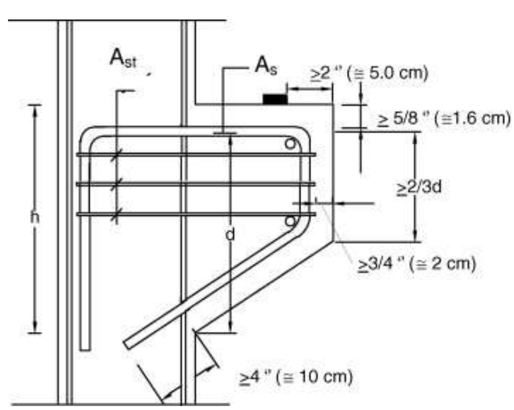


Figure 2.11 Typical requirements for detailing concrete corbel

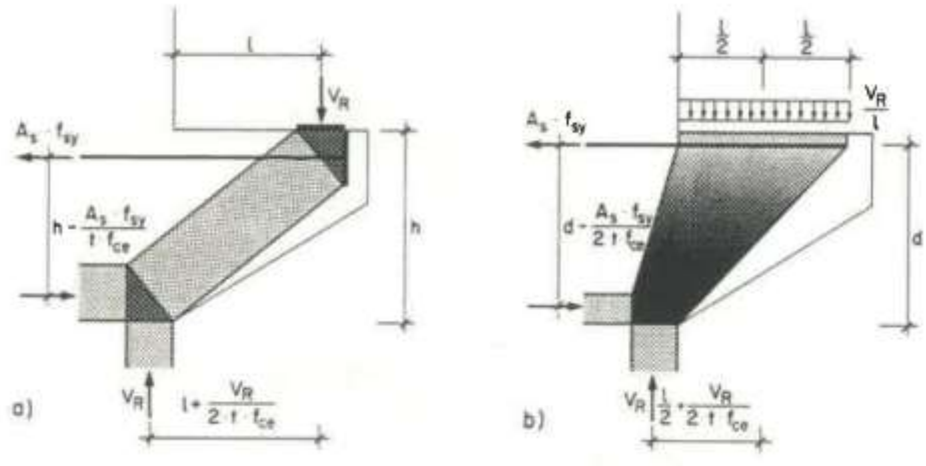


Figure 2.12 Strut-and-tie models for corbel (Muttoni et al., 1997)

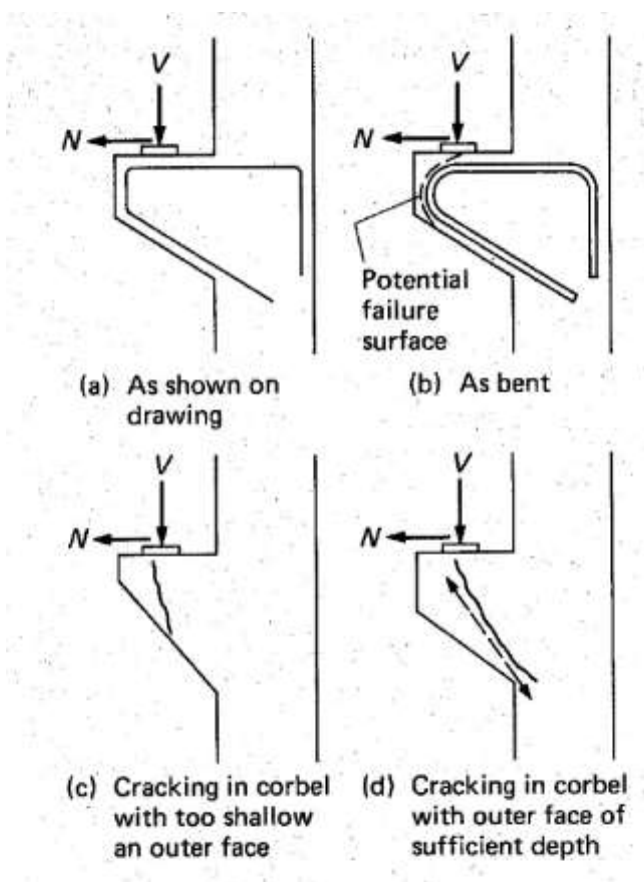


Figure 2.13 Corbel details and possible failure modes (Wang & Salmon, 1992)

2.5.4 Dapped-end beam

Dapped-end beam is the beam with reduced depth at its end. Dapped beam is generally applied for precast beams, bridge girders and so on. The main purpose is to reduce overall height of the structure and to match with the geometry of corbel or bracket for load bearing support. Fig. 2.14 presents truss model for dapped end beam.

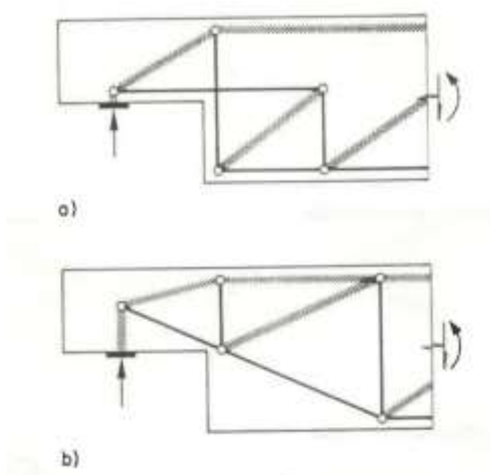


Figure 2.14 Truss model for dapped-end support (Muttoni et al.)

Attention must be paid to the location of vertical and horizontal ties as well as diagonal strut. The horizontal tie must be properly anchored (usually by mechanical welding) to the bearing support to prevent pull out force. The vertical tie acts as hanging bar to carry vertical load of the beam. Diagonal strut usually control the depth of dapped end as well as beam thickness. An example of reinforcement detail for dapped-end beam is shown in Fig. 2.15.

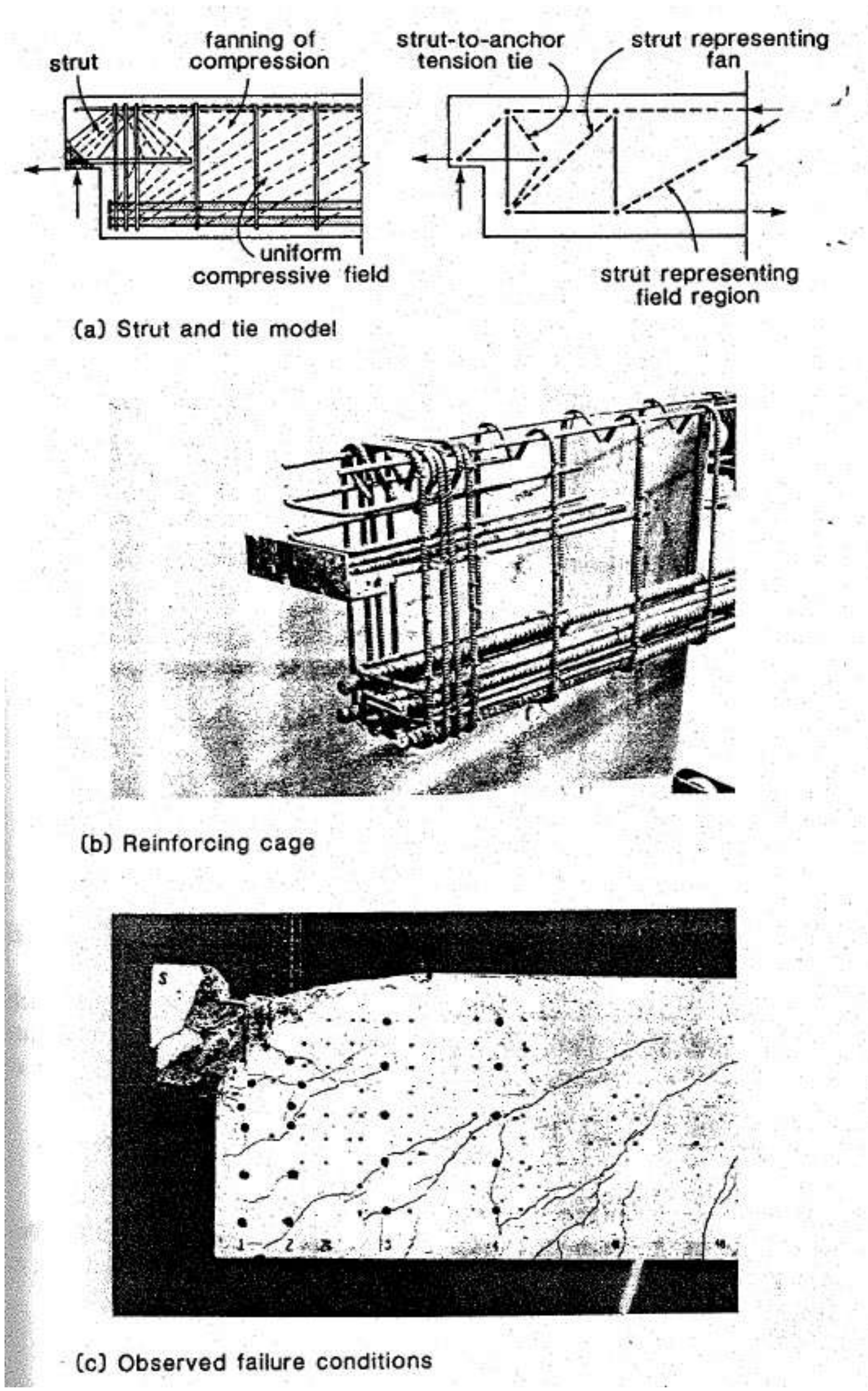


Figure 2.15 Strut and tie model and truss idealization for beams with dapped ends (Collins and Mitchell, 1991)

2.5.5 Pile caps

Pile caps or footings are reinforced concrete elements that transfer loads from columns or walls to the foundation. They are usually thick. Strut-and-tie models of pile caps depend on pile arrangement and load characteristics. Fig. 2.16 present strut-and-tie models of 3-pile footing subjected to axial load and bending moment. The reinforcement details corresponding to strut-and-tie model are also shown. Fig. 2.17 present strut-and-tie model for 4-pile footing with suggested reinforcement details. It should be mentioned that ties must be properly anchored at the boundary of footing in order to develop its yield strength as assumed in the analysis model. The anchorage length is a key factor to dictate concrete covering

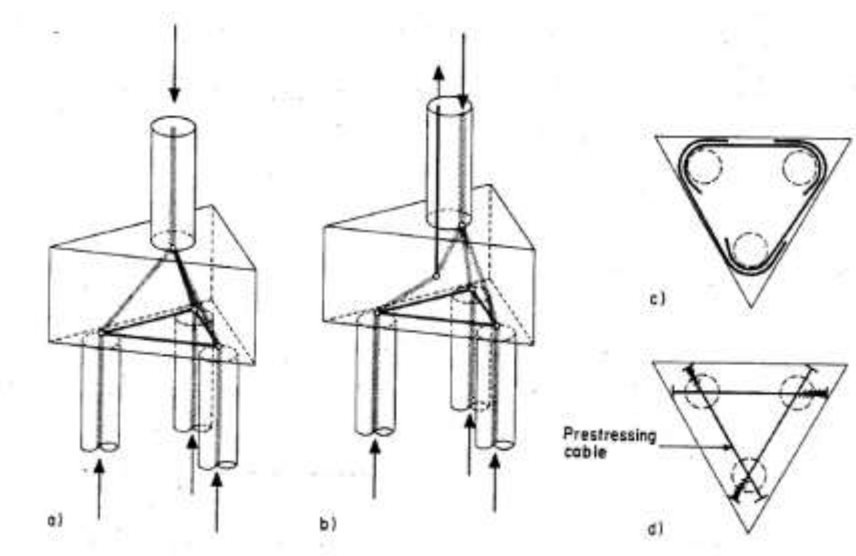


Figure 2.16 Strut-and-tie model of 3-pile footing (Muttoni et al.)

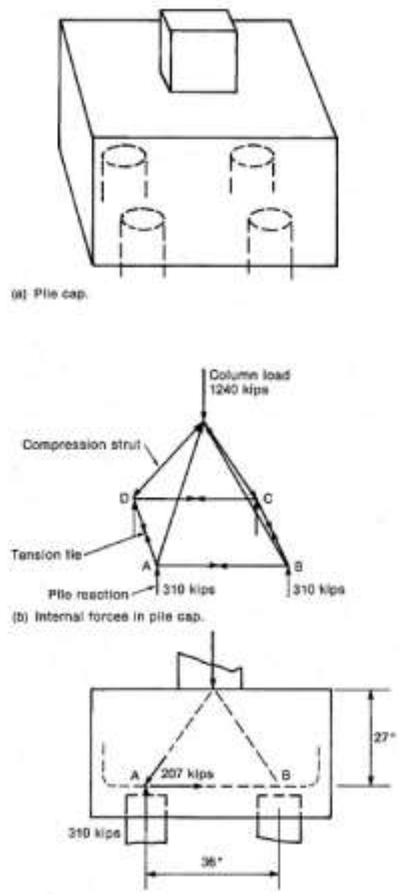


Figure 2.17 Strut-and-tie model of 4-pile footing (MacGregor, 1997)

2.5.6 Joints

2.5.6.1 Beam-column joint

Beam-column joints are subjected to bending moment and shear force from end of beam and column members. Capacity of beam-column joint can be evaluated by using strut-and-tie modeling as shown in Fig. 2.18

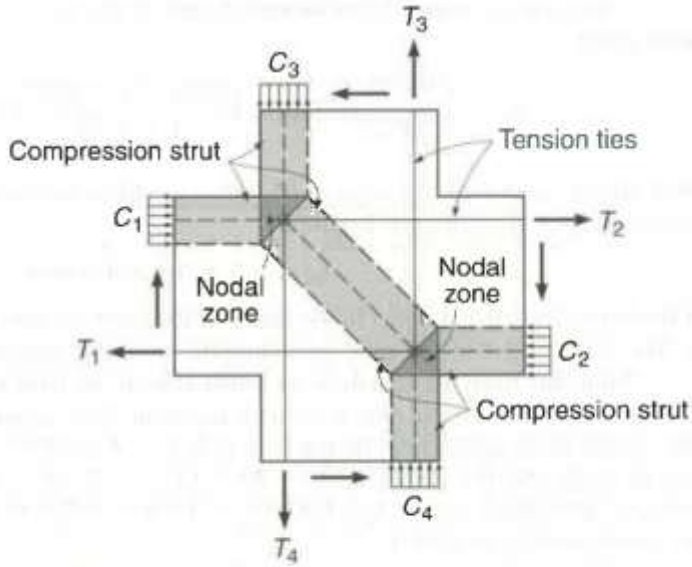


Figure 2.18 Strut-and-tie model for behavior of beam-column joint (Nilson et al., 2010)

2.5.6.2 Beam-to-girder joints

When secondary beam is joined to the main girder, vertical load may introduce diagonal crack near to the joint. Fig. 2.19 presents truss model for beam-to-girder joint. It should be noted that vertical force from the secondary beam is transferred through diagonal strut and vertical tie beam acting as hanging bar. Hanging stirrups shall be provided for complete load path as shown in Fig. 2.19 (b).

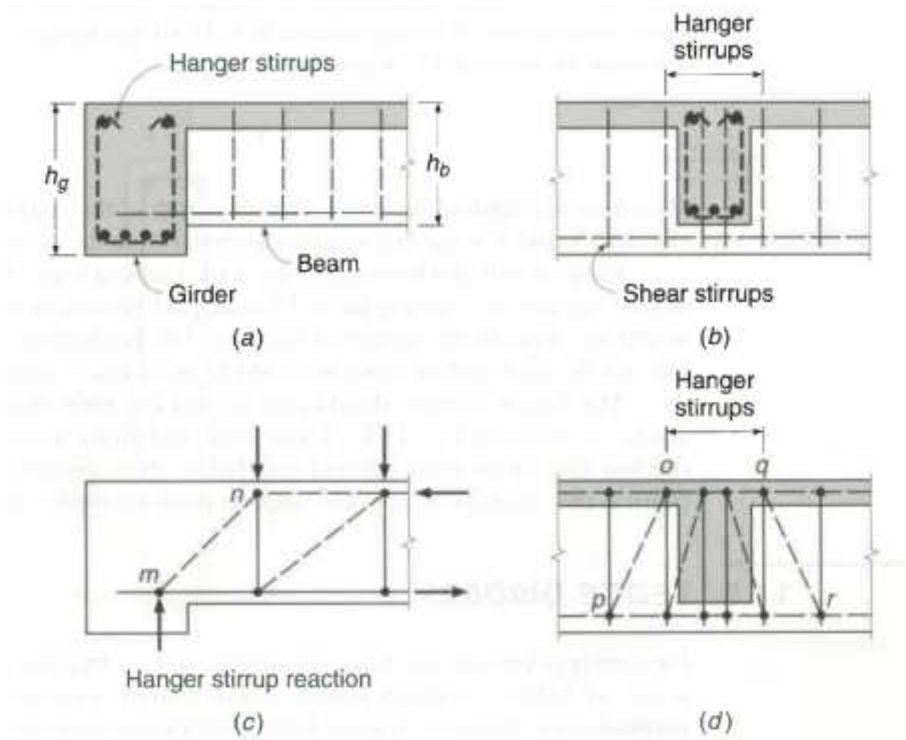


Figure 2.19 Main girder supporting secondary beam (Nilson et al., 2010)

2.5.6.3 Corner joints

Various types of truss model and corresponding reinforcement details are given in Schalach et al. (1987) as shown in Fig. 2.20. It should be mentioned that, those different truss models share the same joint behavior.

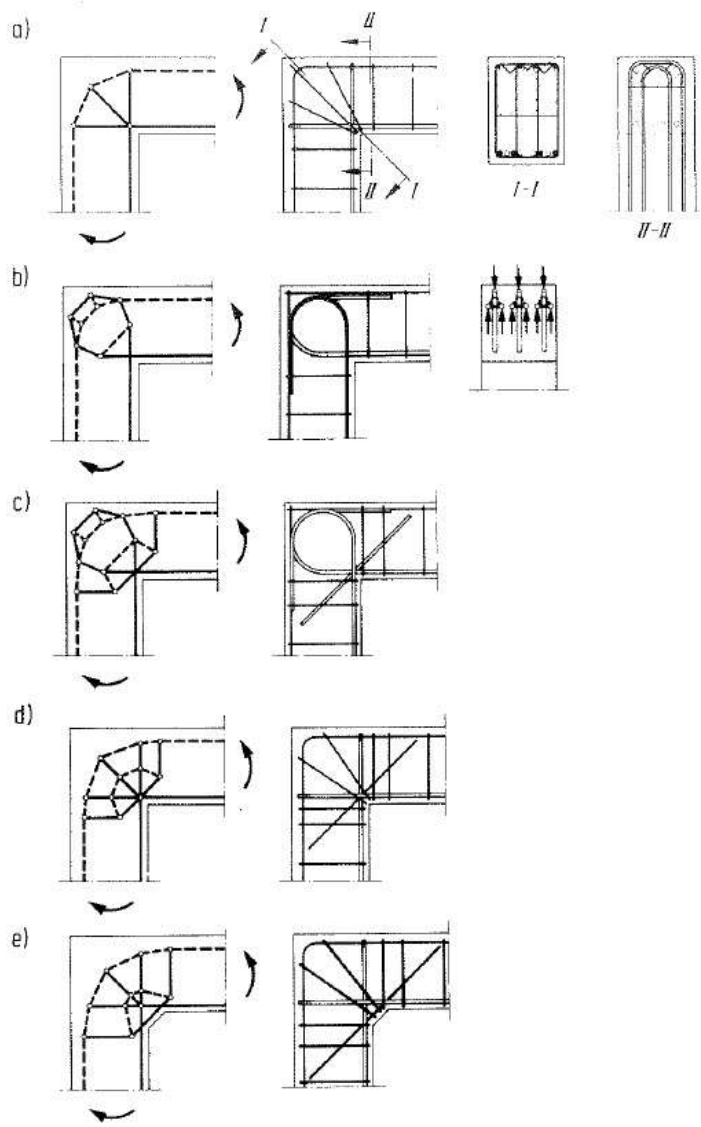


Figure 2.20 Truss models of corner joint and corresponding reinforcement details (Schlaich et al. 1987)

Chapter 3

ACI 318M-08 Appendix A Provisions

Guidance for determining size of struts, nodes and ties has been given in ACI 318M-08 Appendix A. The effective compressive strength of concrete $0.85 f_c'$ is adjusted by a modified factor β to account for the effects of cracks (caused by spreading compressive resultants) and confining reinforcement in struts and the anchorage of ties in nodal zones.

3.1 Strength of compression struts

The effective strength of strut is computed from

$$f_{ce} = 0.85 \beta_s f_c' \tag{3.1}$$

The value of β_s is dependent on shape of idealized strut as well as sufficiency of transverse reinforcement. It accounts for the effects of cracking and confinement within the strut. The values of β_s are summarized in Table 3.1.

Table 3.1

β_s values for strut strength

Condition	β_s
Strut with uniform cross section over its entire length	1.0
Strut with the width at midsection larger than the width at the nodes (bottle-shaped strut) and with reinforcement satisfying transverse requirements	0.75
Strut with the width at midsection larger than the width at the nodes (bottle-shaped strut) and reinforcement not satisfying transverse requirements	$0.60\lambda^{\dagger}$
Struts in tension members or in the tension flange of members	0.40
All other cases, Fig. 10.11	0.60λ

† λ equals 1.0 for normalweight concrete, 0.85 for sand-lightweight concrete, and 0.75 for all-lightweight concrete.

Compression steel may be added to increase the strength of a strut, so that

$$F_{ns} = f_{ce} A_{cs} + A_s' f_s' \tag{3.2}$$

where f_s' is based on the strain in the concrete at peak stress. For grades 280 and 420 Mpa reinforcement, f_s' may be assumed as f_y. If compression rebar is accounted for calculation of strength, ACI code A3.5 requires that the compression bar must be properly anchored, oriented parallel to the axis of strut, located within the strut, and enclosed by ties or spirals, as required for columns.

Where that lateral spreading of compression field is possible, ACI code A3.3 allows strut to be idealized as bottle-shaped (Fig. 2.1). Spreading of compressive stress can cause crack due to longitudinal splitting. Because of this, bottle-shaped struts are weaker than prismatic struts. Transverse reinforcement shall be provided to control longitudinal splitting. For $f_c' \le 42$ Mpa, the ACI code considers the transverse reinforcement requirement to be satisfied if the strut is crossed by layers of reinforcement that satisfy

$$\sum \frac{A_{si}}{b_s s_i} \sin \alpha_i \geq 0.003 \tag{3.3}$$

where $\frac{A_{si}}{b_s s_i}$ is the content of the rebar in layer *i*. The multiplication by $\sin \alpha_i$ is to account for the component of reinforcement transverse to the longitudinal axis of strut.

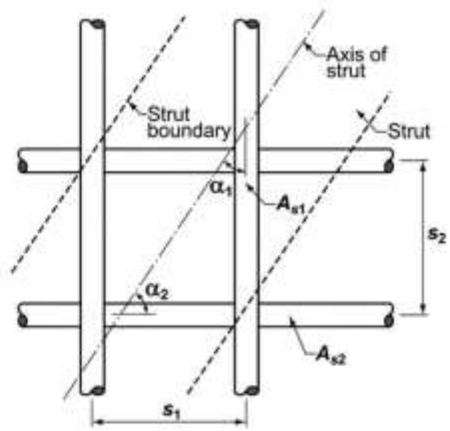


Figure 3.1 Reinforcement crossing at strut

3.2 Strength of nodal zones

Once nodal zone dimensions are defined, the nominal strength of nodal zone can be computed from the following equation

$$F_{nn} = f_{ce} A_{nz} \tag{3.4}$$

where A_{nz} is the area of the face of nodal zone taken perpendicular to the line of action of the force from strut or tie

f_{ce} is the effective compressive strength of the concrete in the nodal zone, given by

$$f_{ce} = 0.85 \beta_n f_c' \tag{3.5}$$

The value of β_n is dependent on type of nodal zone as shown in Table 3.2. For CCC node, the value of β_n is maximized where the β_n value are penalized in CTT or TTT nodes.

Table 3.2

β_n values for node strength

Nodal Zone Condition	Classification	β_n
Bounded by struts or bearing area	*C-C-C*	1.0
Anchoring one tie	*C-C-T*	0.80
Anchoring two or more ties	*C-T-T* or *T-T-T*	0.60

3.3 Strength of ties

The nominal strength of ties F_{nt} is contributed by the strengths of reinforcing steel and prestressing steel within the tie. Concrete in tension does not contribute any strength to ties.

$$F_{nt} = A_s f_y + A_p \left(f_{pe} + \Delta f_p \right) \tag{3.6}$$

where A = area of reinforcing steel

f_y = yield strength of reinforcing steel

A_p = area of prestressing steel, if any

f_{pe} = effective stress in prestressing steel (initial stress)

Δf_p = increase in prestressing steel stress due to factored load increment

Sum of $f_{pe} + \Delta f_p$ must be less than the yield stress of the prestressing reinforcement f_{py}. The value of Δf_p may be found by analysis. ACI Code A.4.1 allows a value 420 Mpa to be used for bonded tendons and 70 Mpa to be used for unbounded tendons.

Effective width of tie w_t depends on the distribution of tie reinforcement. It can be taken as the width of anchor plate. The practical upper limit for tie width $w_{t,\max}$ is equal to the width corresponding to the width of hydrostatic nodal zone, given as

$$w_{t,\max} = \frac{F_{nt}}{b_s f_{ce}} \tag{3.7}$$

where f_{ce} is the effective nodal zone compressive stress and b_s is thickness of strut

Ties must be properly anchored before they leave the nodal zone at point defined by the centroid of the bars in the tie and the extension of the outlines of either the strut or the bearing area as shown in Fig. 2.5. If the combined lengths of nodal zone and extended nodal zone are inadequate to provide for development of the reinforcement, additional anchorage may be obtained by extending the reinforcement beyond the nodal zone , using 90^{o} hooks, or by using mechanical anchor.

3.4 Shear requirements for deep beam

Deep beams are defined as the beam having clear spans to total member depth (L/h) less than or equal to 4 (Fig. 3.2 (a)). Also, for the beam with concentrated load placed within twice of total beam depth from support (Fig. 3.2 (b)) is considered as deep beam. ACI code 11.8.3 specifies that the nominal shear in deep beam may not exceed $0.83\sqrt{f_c'}b_w d$, where b_w is the width of the web and d is effective depth. ACI Code 11.7.4 and 1.7.5 provide minimum steel requirements for horizontal and vertical reinforcement within a deep beam. The minimum reinforcement perpendicular to a span is

$$A_v \geq 0.0025 b_w s \tag{3.8}$$

where s is the spacing of the reinforcement. The minimum reinforcement parallel to a span is

$$A_h \geq 0.0015 b_w s_2 \tag{3.9}$$

where s_2 is the spacing of the reinforcement perpendicular to the longitudinal reinforcement. Spacing s and s_2 may not exceed $d/5$ or 300mm (whatever less). ACI Code 11.7.6 allows Eq. (3.3) to be used instead of Eqs. (3.8) and (3.9). For strut-and-tie models, b_w is equal to the thickness of the element b.

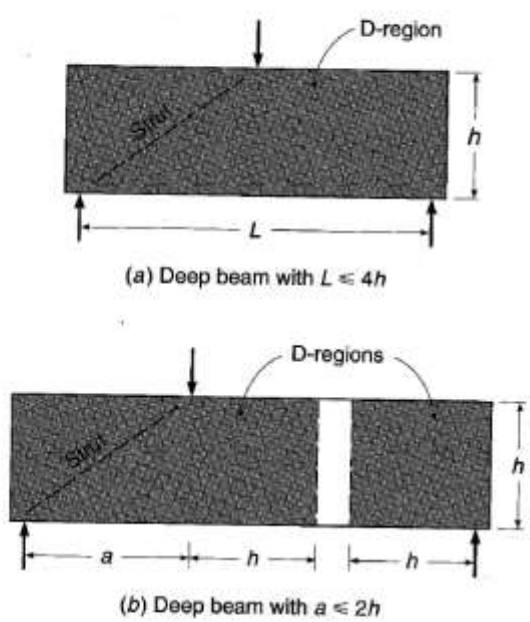


Figure 3.2 Deep beam D-regions

Chapter 4

STM Software and Applications

4.1 Fachwerk & Fachwerk3D

Fachwerk and Fachwerk3D are Java-based applications developed by Adrian Vontobel (2010) are software for 2D and 3D STM analysis, respectively. The programs base on the principle of minimum complimentary strain energy (Castigliano Theorem), so it can be applied without knowing information of element stiffness. They are under GNU open source project that means they are freely distributed. Fachwerk and Fachwerk3D can be downloaded at http://fachwerk.sourceforge.net/index_en.html. The main features of Fachwerk and Fachwerk3D are listed as the following:

Analysis

- Calculation of forces in strut and tie elements
- Indicate failure mechanism when the program detect any static inadmissible conditions
- Allow user to input specified member forces for indeterminate model, i.e. forces along prestressed concrete tendon
- Allow user to specify force to any ties, i.e. yield force to predict limit load behavior
- The program employs only equilibrium condition without considering element stiffness as required by STM methodology

- Allow node relocation to satisfy assumed target element forces. This is benefit to sensitivity analysis of STM topology.
- Indicate degree of static indeterminacy at real time

GUI

- Interactive graphical user interface with edit, pan, and zoom capabilities
- Calculation report can be printed out

Some screenshots from Fachwerk and Fachwerk3D are shown in Fig. 4.1 and 4.2, respectively.

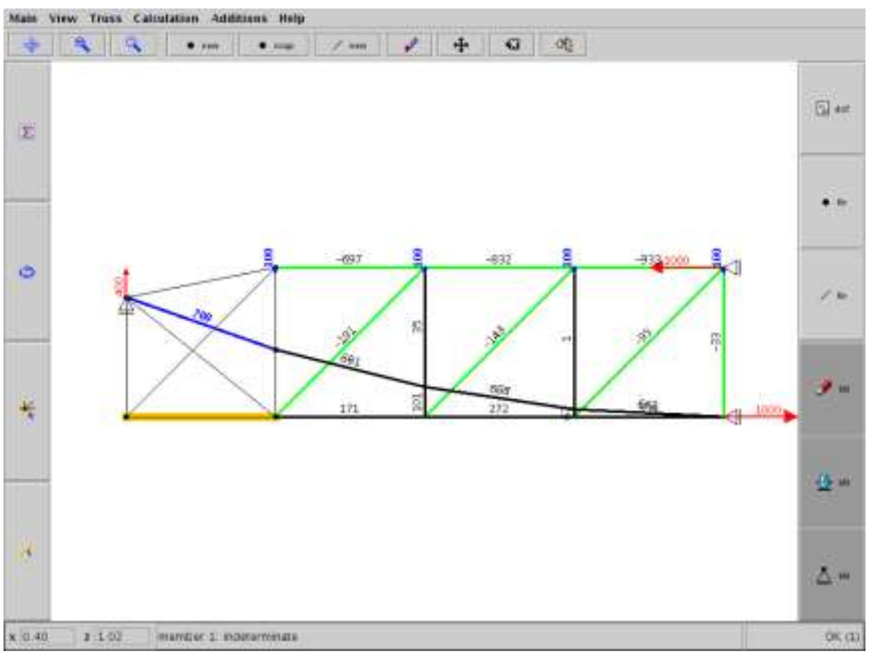


Figure 4.1 A screenshot from Fachwerk

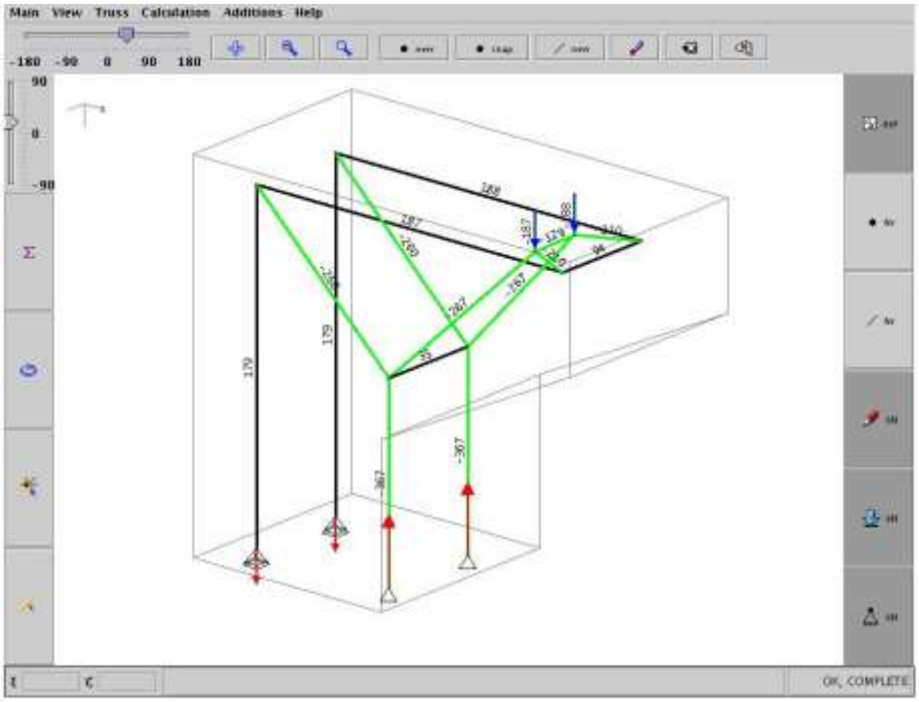


Figure 4.2 A screenshot from Fachwerk3D

At the bottom line, Fachwerk and Facwerk3D neither check element strength nor provide nodal zone dimensioning. The software aims to structural engineers who just want a portable modeling and analysis tools for 2D and 3D STM. The software serves very well for such objective.

4.2 CAST

CAST stands for *Computer aided for Strut-and-Tie* design tools. It is only an existing windows-based application for STM with rich of graphical user interfaces. CAST has been developed by the University of Illinois since 2001 (Kuchma & Tjhin, 2001). It was used to serve students and practicing engineers to grasp the concept and to try various idealized strut-and-tie models with ease. Main features of CAST can are listed as shown in Fig. 4.3.

CAST components comprise of nodes and elements. Elements represent struts or ties meeting at nodes. For particular cases, stabilizer is necessary to make the model numerically stable. The stabilizer is a strut with zero force as shown in Fig. 4.4. Three types of boundary conditions can be applied at exterior nodes: plate, point loads, and supports. Body force and support can also be applied to interior nodes within D-region. The model must be restrained to prevent rigid body movement. In CAST, positive value of forces is point outward from node and vice versa. Since the direction of point loads on the boundary will follow the axis of the strut, no element connectivity is allowed on boundary. The following procedure is necessary for performing strut-and-tie analysis by CAST:

Graphical User Interface

- Define D-Region boundaries graphically
- Define strut-and-tie models graphically and numerically
- Guidelines, grid points, and snap tools for accurate dimensioning
- Toolbars and dialog boxes for defining common elements, such as the imposed loadings, material properties, supports, and bearing plates
- Templates for common truss models
- Standard drawing and editing features, such as cut, copy, paste, delete, undo, redo, move, find, glue, and unglue
- Easy access to member information by mouse right button click
- Display of structural model, including reinforcement positions, labels, and areas, in multiple windows with various zooming and panning options
- Context sensitive online help

Analysis Types

- Design calculations (evaluation of truss member forces and truss member dimensions)
- Detailed analysis of nodal zones
- Simple truss capacity prediction
- Load-deformation analysis

Truss Solution

- Truss analysis and stress check at command
- Statically indeterminate trusses
- Truss stability checked with warning messages
- Illustration of member forces and identification of member characteristics for dimensioning
- Pushover analysis per user-defined stress-strain relationships of struts and ties with options to use load-controlled or displacement-controlled

Multiple Load Cases and/or Strut-and-tie Models

- Organize multiple load cases and/or strut-and-tie models in one database
- Present multiple load cases and/or strut-and-tie models in multiple windows

Truss Member Dimensioning

- Manual or automatic selection of effective widths of struts
- Single or multiple layers of tie reinforcement
- Checks of bearing plate stresses
- Quick check of the adequacy of strut-and-tie model components through color identification
- Support explicitly ACI Code requirements

Output Features

- Graphical and tabular output
- Create customized text input echo and/or output files
- Create design summaries including graphics in Microsoft Word Document format
- Create input echo and/or output tables in Microsoft Excel format
- Create graphics printout with print preview
- Export graphics into AutoCAD DXF and Bitmap files
- Create response quantity plots obtained from load-deformation analysis in Microsoft Excel format
- Display response history obtained from load-deformation analysis

Miscellaneous

- Support SI, US customary, and old metric units interchangeable during the design
- Support user-defined and ASTM Standard Reinforcing Bars (ASTM A615/A615M)
- Unit converter for helping convert physical quantities from one unit to another
- Built-in calculator for editable text boxes
- Customizable windows and toolbars

Figure 4.3 Current features of CAST (Version 0.9.11/ 01.26.2004)

1. Specify general properties of the STM (Fig. 4.4). They are D-region thickness, compressive strength of concrete (f_c'), tensile strength of concrete (f_{ct}) and non-prestressed reinforcement yield strength (f_y)

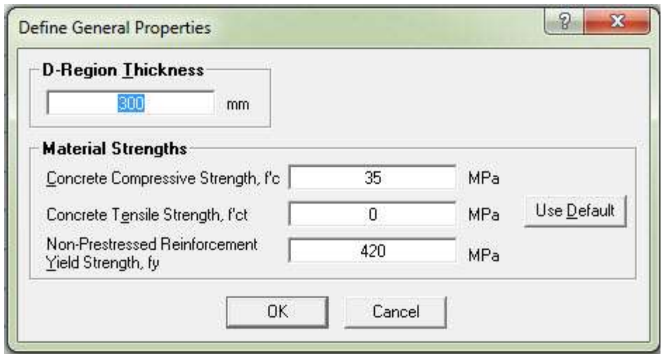


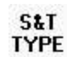
Figure 4.4 General properties window

2. Create the boundary of the D-region by using  command. Interior boundary (voids) can be constructed by using  command. The boundary nodes can be relocated by using  command.

3. Define nodes by using  command and elements by using  command. After construction, nodes can be relocated by using  command.

4. Assign boundary conditions at nodes by using  command for bearing support,  for nodal forces or supports. Body forces or supports can be assigned to interior nodes by using  command.

5. Define and assign struts types (ACI prismatic strut , ACI bottle-shaped strut with reinforcement , ACI bottle-shaped strut without reinforcement , or ACI strut in tension member ). Moreover, user-defined struts are allowed by using 

command. Tie can be defined and assigned by using 

command.

6. Specify and assign relative stiffness and width of struts and ties by using 

command (Fig. 4.5).

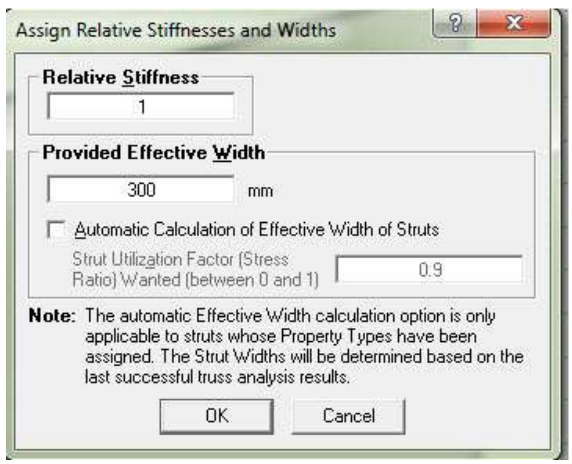


Figure 4.5 Input window for assigning relative stiffnesses and widths of struts and ties

7. Perform analysis run by  command. By default, the program determines nodal zone dimension and perform checking of nodal stresses.

8. Review the results for forces in struts, tie, as well as utilization stress ratio. If the stress ratio is less than 1.0, the result is acceptable. Otherwise, width & thickness of struts must be increased or the amount of rebar in ties must be increased. The maximum thickness of struts and ties is controlled by the thickness of D-region.

Some screen shots of CAST are shown in Figs. 4.6 to 4.8.

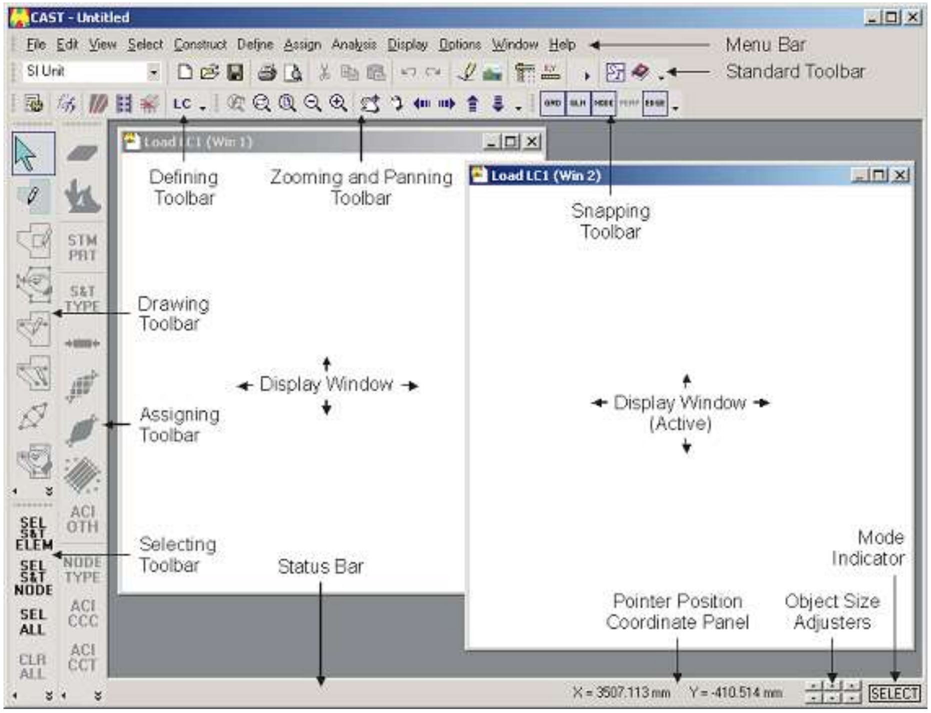


Figure 4.6 CAST working environment

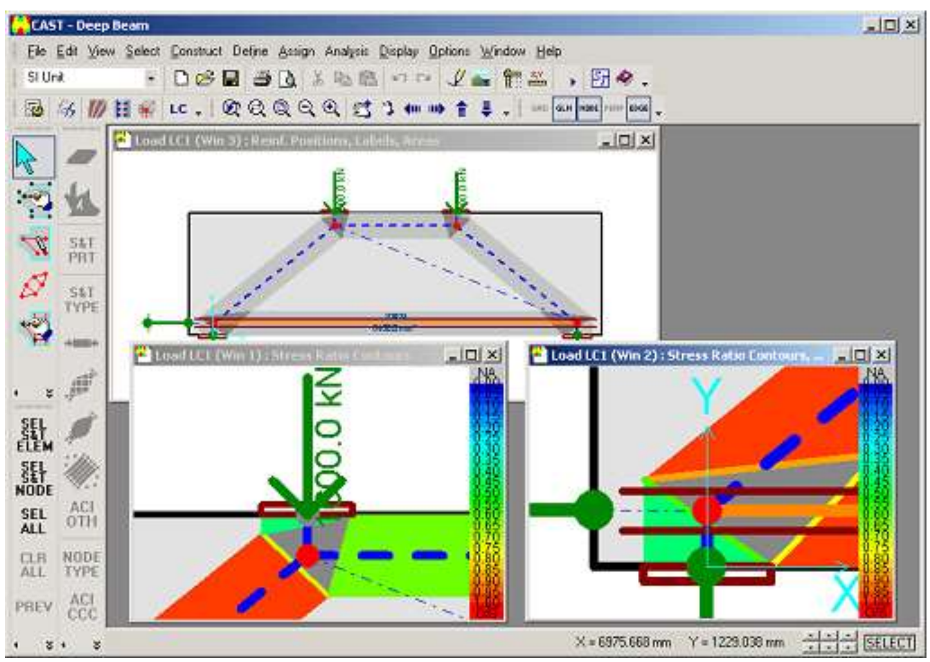


Figure 4.7 Deep beam under two-point load

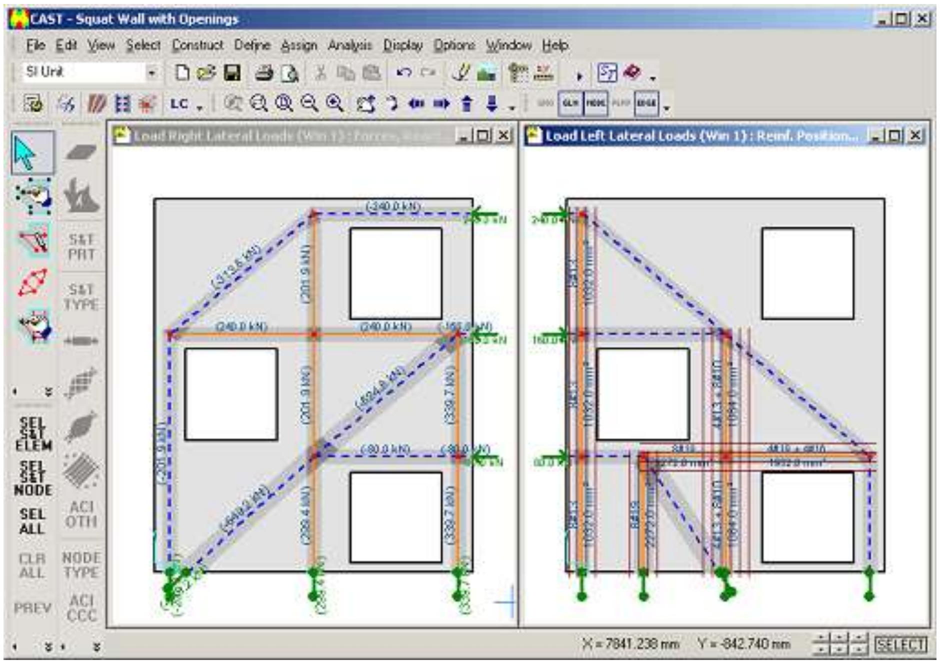


Figure 4.8 Non-slender wall with openings

Chapter 5

Worked Examples with CAST

This chapter demonstrates the strut-and-tie method for solving two practical problems. The first example is double corbel problem and the second is deep beam problem. Although, it is possible to solve those problems by hand calculation, we use CAST in order to give an impression of computer aided design tool for strut-and-tie modeling.

5.1 Double Corbel

This example is obtained from CAST's tutorial page†. It presents analysis of double corbel problem using strut-and-tie method. The geometry of the corbel is shown in Fig. 5.1. By this example, CAST is used to construct the truss model, to analyse and to determine stresses in struts, ties and nodal zones. The program needs user to input almost all important parameters, such as material properties, width and thickness of strut and ties, number of rebar in tie, as well as node classifications. The program automatically determines nodal zone dimension based on hydrostatic concept or prescribed nodal zone dimension. For the given problem, $b = 600$ mm, $f_c' = 35$ Mpa, $f_{ct} = 0$ and $f_y = 420$ Mpa. The following procedure shall be followed to complete this example:

† (http://dankuchma.com/stm/CAST/tutorials/dcorbel/index.htm)

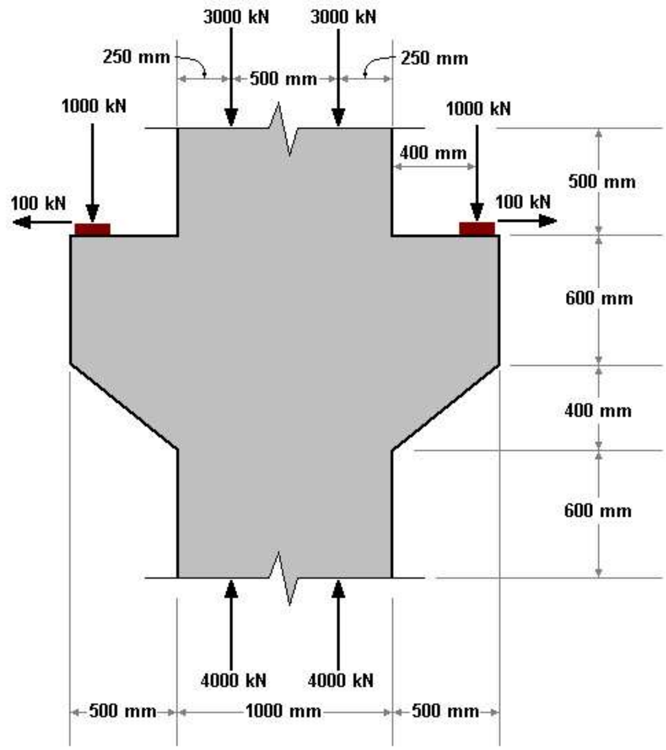


Figure 5.1 The geometry and loadings of the corbel

Step 1: Construct the outer boundary

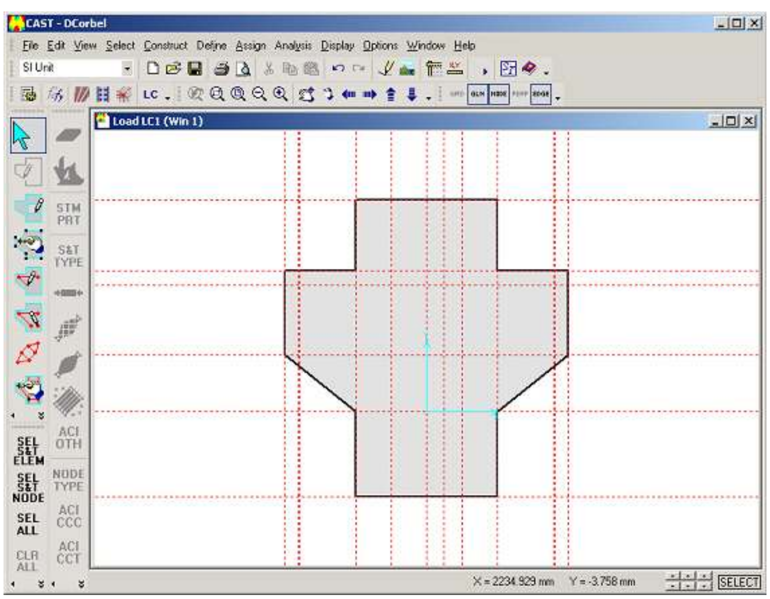


Figure 5.2 Outer boundary with guidelines

Step 2: Construct strut-and-tie model

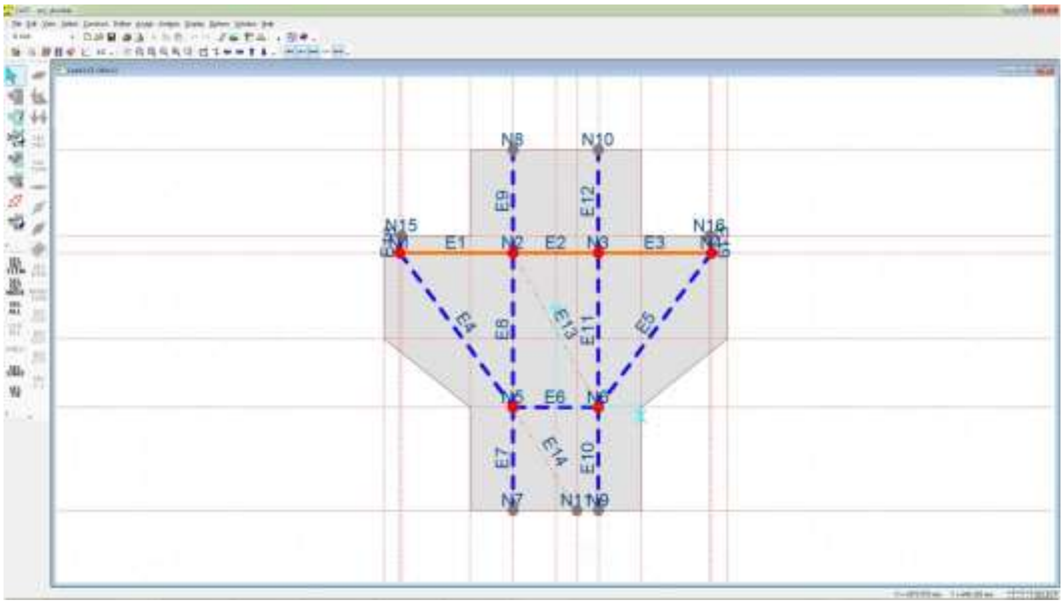


Figure 5.3 Node and elements

Step 3: Apply loads and boundary supports

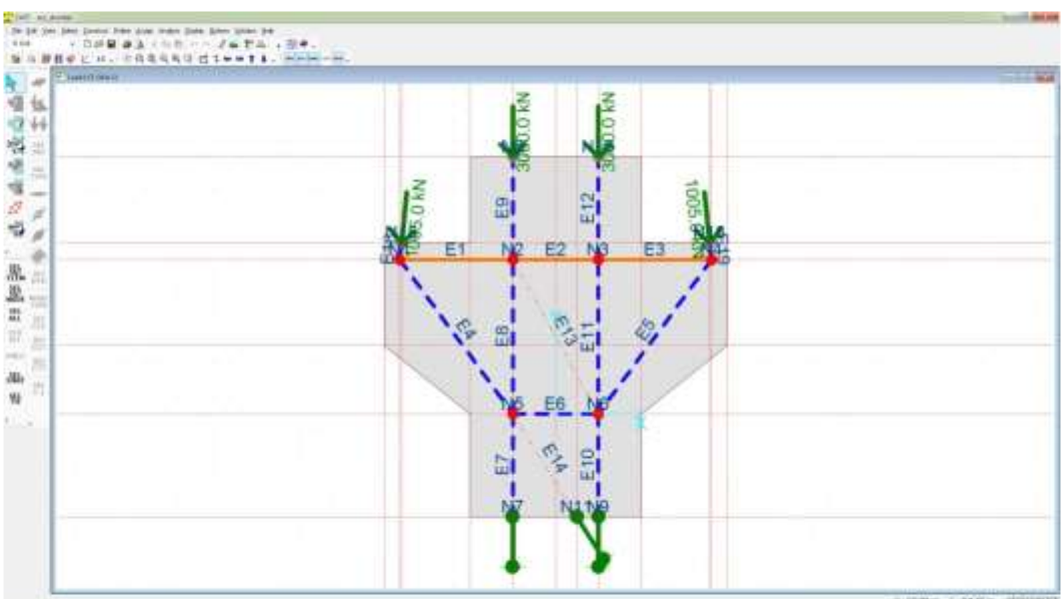


Figure 5.4 Model with loads and supports

At this stage, it should be mentioned that CAST can only accept resultant forces. The element E18 and E19 must be inclined in to the direction of resultant forces which can be determined from $\tan\theta = F_x / F_y$ where θ is measure from the vertical axis.

Step 4: Define all struts as ACI prismatic strut except E4 and E5 as bottle-shaped struts. Tie E1, E2 and E3 are defined as non-prestressed rebar (6-M25) as shown in Fig. 5.5

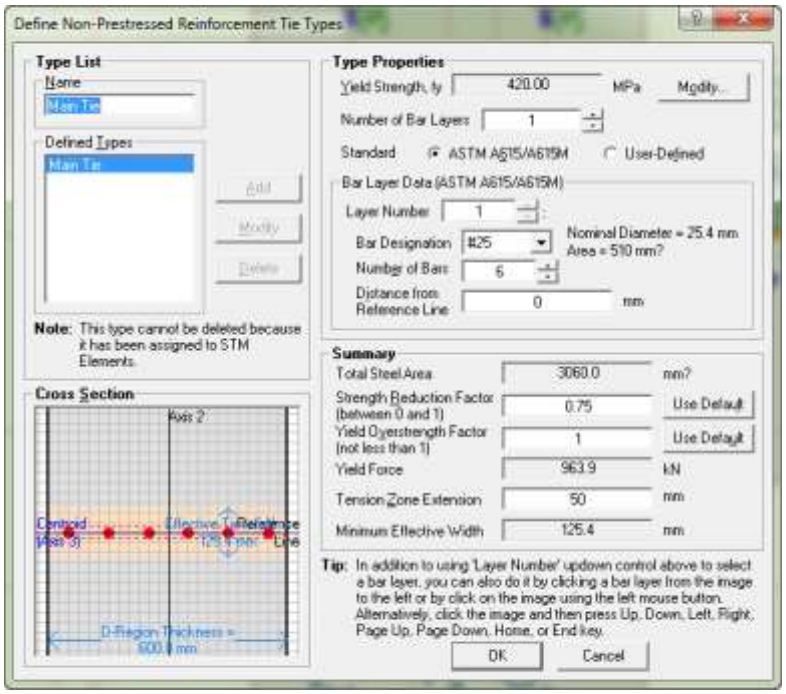


Figure 5.5 Properties of ties

Step 5: Assign width of struts and ties

In this example, width of ties E1, E2 and E3 are 200 mm, width of struts E7-E12 are 300 mm, and width of the rest struts are 150 mm.

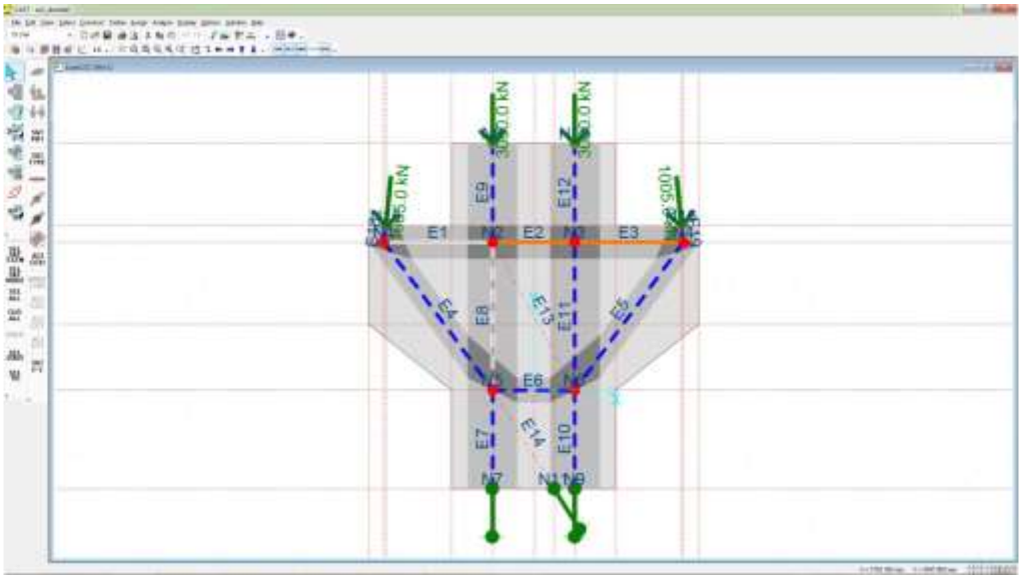


Figure 5.6 Strut-and-tie dimension

Step 6: Assign node types

The user must manually assign node type to each nodes in the model. In this example, we know that N5 and N6 are CCC nodes; N1 and N4 are CCT nodes. It should be noted that only nodes to be checked by the program are needed to be specified. The program will determine the maximum stress from the defined node type.

Step 7: Run analysis and check results

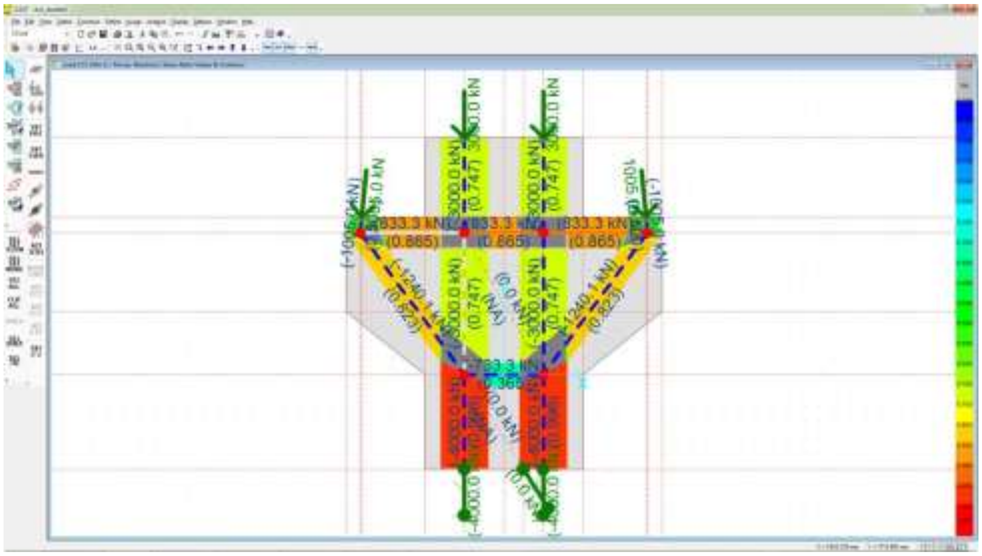


Figure 5.7 Analysis results for forces in truss model and utilized stress ratio

According to the results, there are two elements that have no internal force. They are called by the developer as *stabilizer*. If such elements, are removed before the analysis to be run, the error message indicating model instability will occur and the analysis cannot be proceeded.

Step 8: Check nodal zone stress

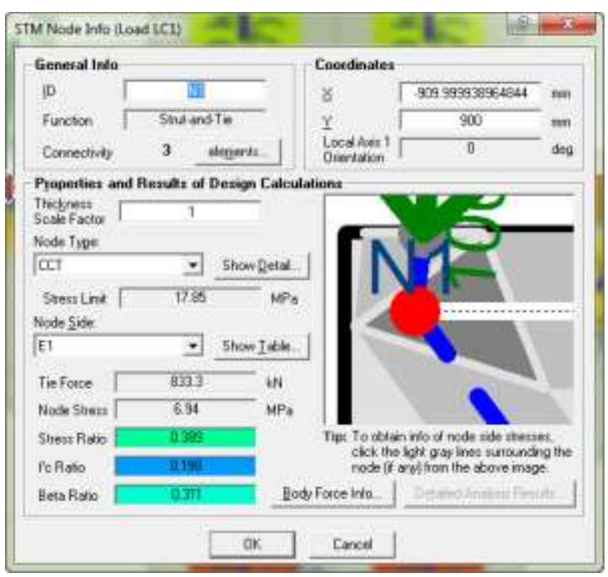


Figure 5.8 Nodal zone stress check

At this stage, only nodes with pre-defined node type contain the results of nodal zone stress check. By right clicking at node, the window as shown in Fig. 5.8 will appear. It should be noted that, the program construct nodal zone dimension based on the known strut and tie dimensions. It simply computes nodal stress from the computed forces in the corresponding struts and ties coincident at node divided by their cross dimension area. For example, at node N1, the face perpendicular to tie, the stress at node is computed as

$$\sigma_{N1} = \frac{833.3 \times 10^3}{200 \times 600} = 6.94 \text{ Mpa}$$

Stress limit for CCT node is $\phi f_{ce} = 0.75 \times 0.85 \times 0.80 \times 35 = 17.85$ Mpa

Utilized stress ratio is 6.94/17.85 = 0.388

f_c' ratio is 6.94/35.0 = 0.198

$$\beta = \frac{f_{ce}}{\phi 0.85 f_c'} = \frac{6.94}{0.75 \times 0.85 \times 35} = 0.311$$

Beta ratio is a measure of perpendicular nodal stress to allowable compressive stress.

5.2 Deep Beam

Another example is deep beam subjected to concentrated load. This example is adopted from a paper of Wight and Parra-Montesinos (2003). The model of deep beam in the paper can be shown in Fig. 5.9.

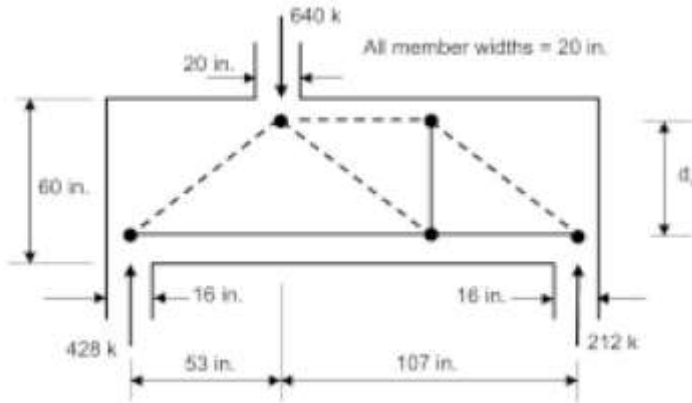


Figure 5.9 Deep beam model (Wight and Parra-Montesinos, 2003)

For this example, we set up the general properties of the problem as shown in Fig. 5.10.

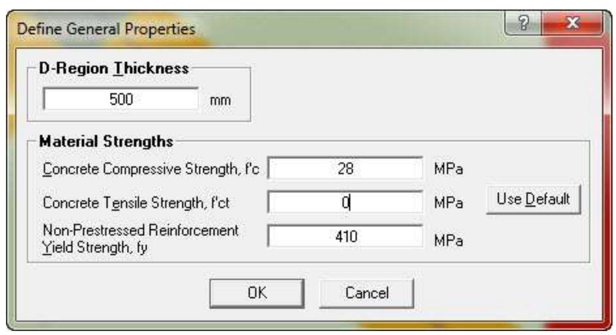


Figure 5.10 General properties of the problem

The following procedure should be followed in order to complete this example:

Step 1: Construct the outer boundary

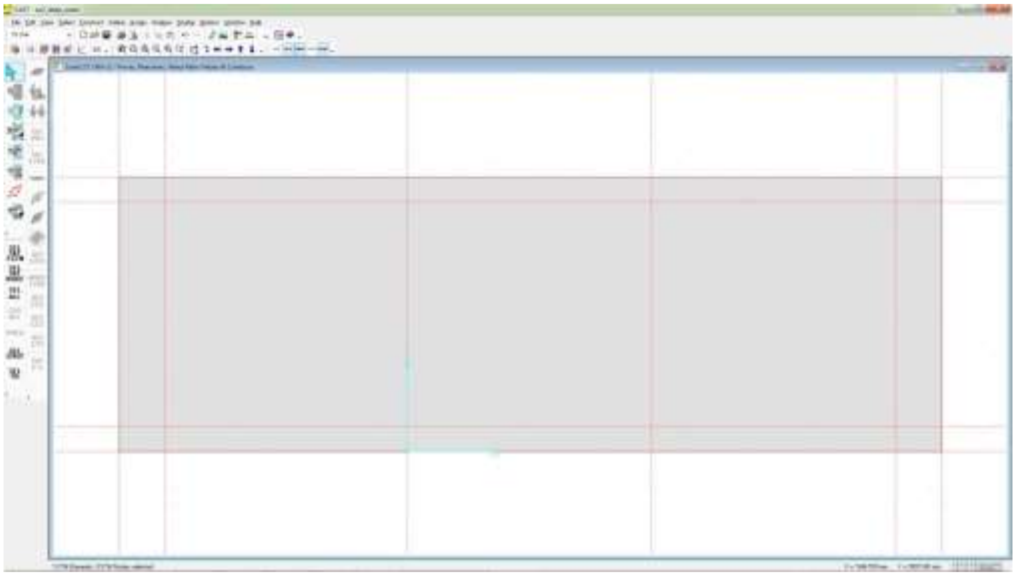


Figure 5.11 Outer boundary

Step 2: Construct truss model

At this stage it is assumed that the depth of truss model is equal to 1245 mm. The truss model can be constructed as shown in Fig. 5.12. Nodal coordinates of the model is shown in Fig. 5.13.

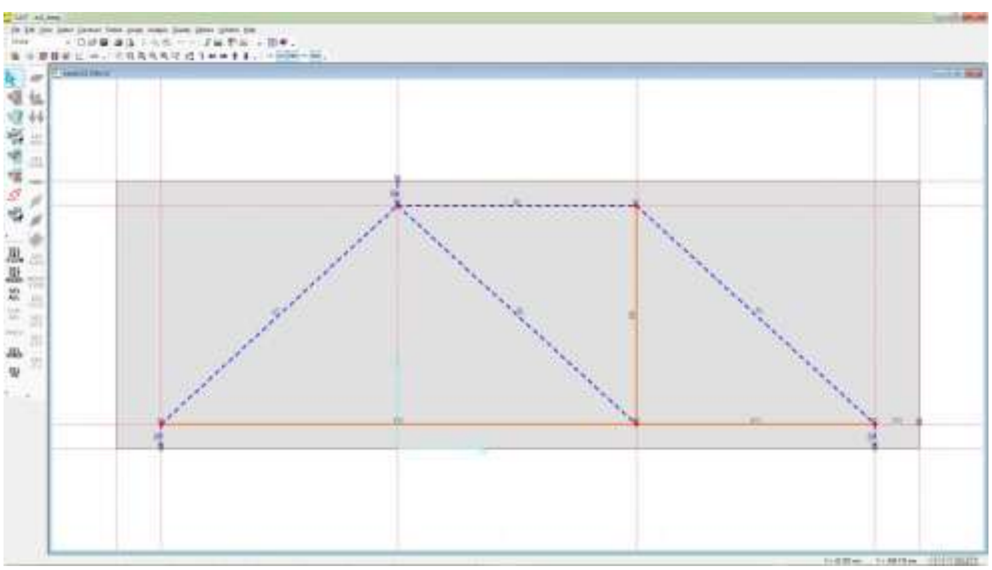


Figure 5.12 Truss model geometry

Row #	Node ID	Function	X (mm)	Y (mm)	Direction (deg.)
1	N1	Strut-and-Tie	-1345	140	0
2	N2	Strut-and-Tie	0	1385	0
3	N3	Strut-and-Tie	2718	140	0
4	N4	Load/Support	2718	0	180
5	N5	Load/Support	-1345	0	180
6	N6	Strut-and-Tie	1359	140	0
7	N7	Strut-and-Tie	1359	1385	0
8	N8	Load/Support	0	1525	0
9	N9	Load/Support	2972	140	-90

Figure 5.13 Nodal coordinates

It should be mentioned that stabilizer element E10 is added to the model to retain numerical stability and enable the program to solve the problem properly.

Step 3: Apply loads and boundary supports

Bearing supports (depth $\times$ width = 500 $\times$ 500 mm) are assigned to N1, N2 and N3, respectively. Vertical load -2909 kN is applied at N2.

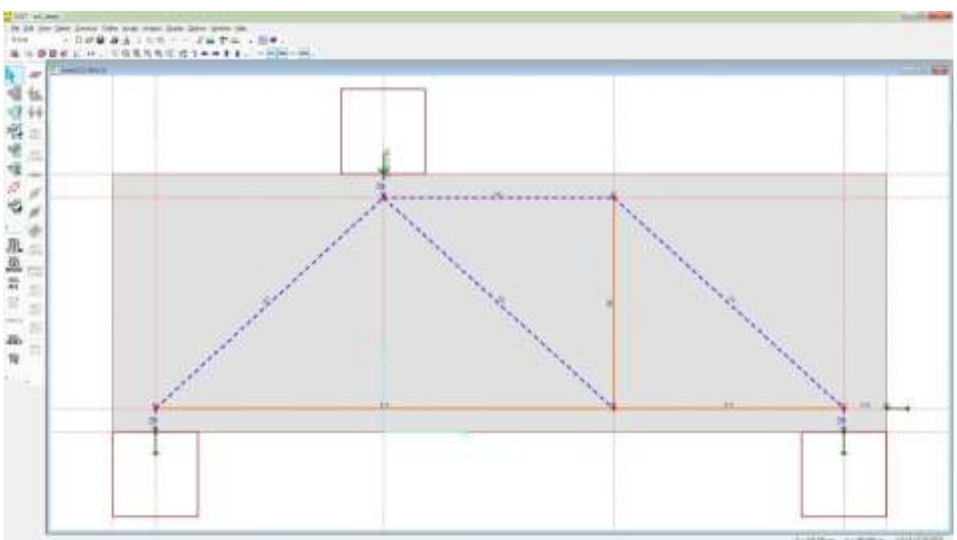


Figure 5.14 Load and boundary supports

Step 4: Define strut types and tie properties

Since lateral spreading of compression field is feasible, the inclined struts E1, E5, and E7 are defined as bottled-shape strut, while strut E8 is defined as prismatic strut.

Step 5: Assign width of struts and ties

CAST has ability to compute the width of struts based on specified utilized stress ratio. We shall demonstrate this feature in this example. Setting stress ratio as 0.90, we do not prescribe the width of the struts at this stage, but ask the program to compute for them.

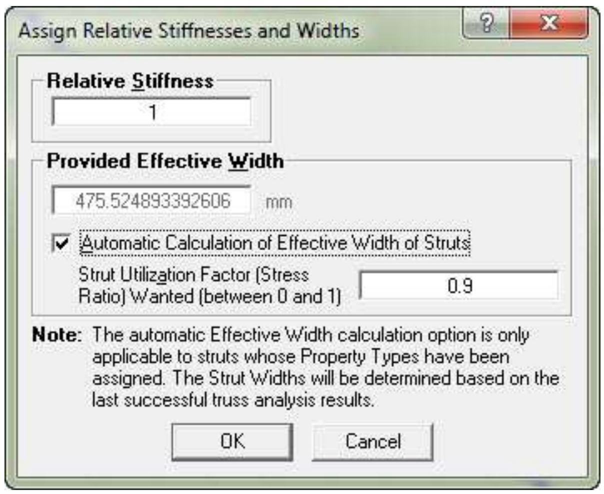


Figure 5.15 Let the program determine width of strut

The main horizontal tie is defined as shown in Fig. 5.16, while the vertical tie is defined as in Fig. 5.17. It should be noticed that the width of both ties are calculated from the depth of outer layers plus of extended effective tensile zone. Width of ties does not affect to its strength, but it affects to stress in nodal zone. Modification of rebar arrangement may be necessary if stress in nodal zone is greater than the allowable value.

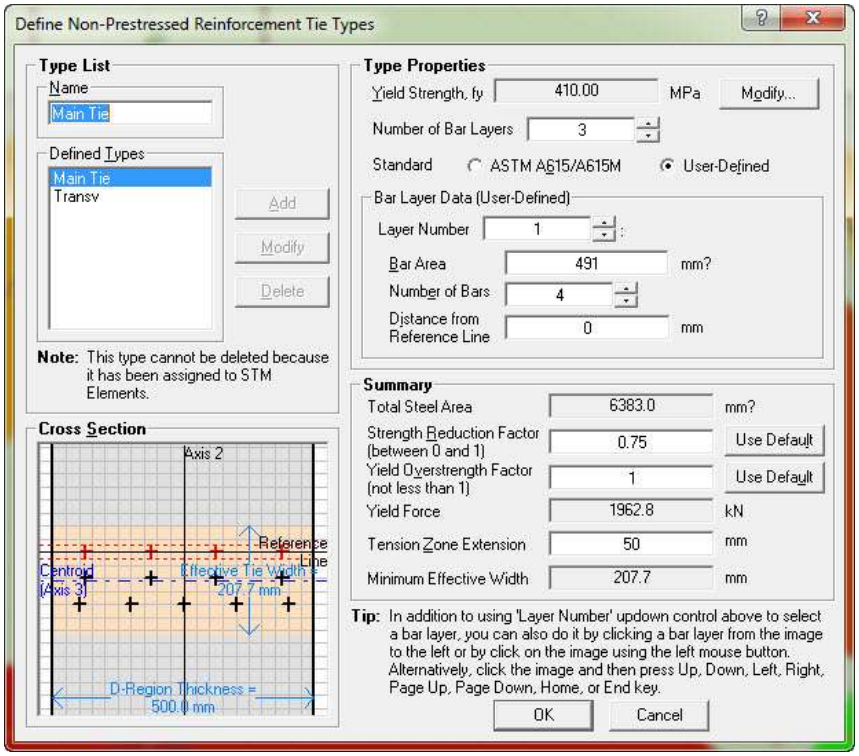


Figure 5.16 Properties of the main tie

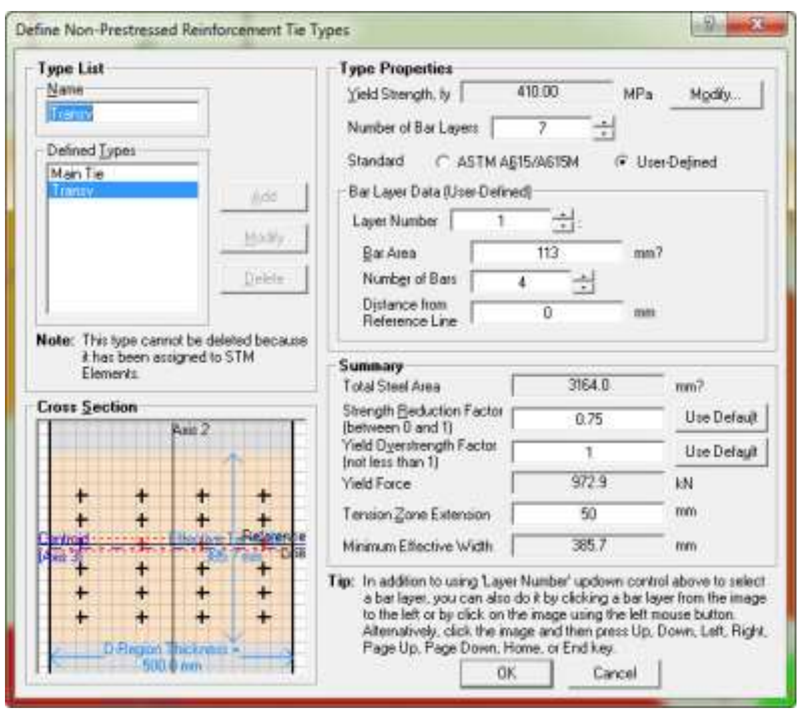


Figure 5.17 Properties of the transverse tie

Step 6: Assign node types

Assign ACI CCC node to N2 and ACI CCT node to N1, N3, and N7

Step 7: Run analysis and check results

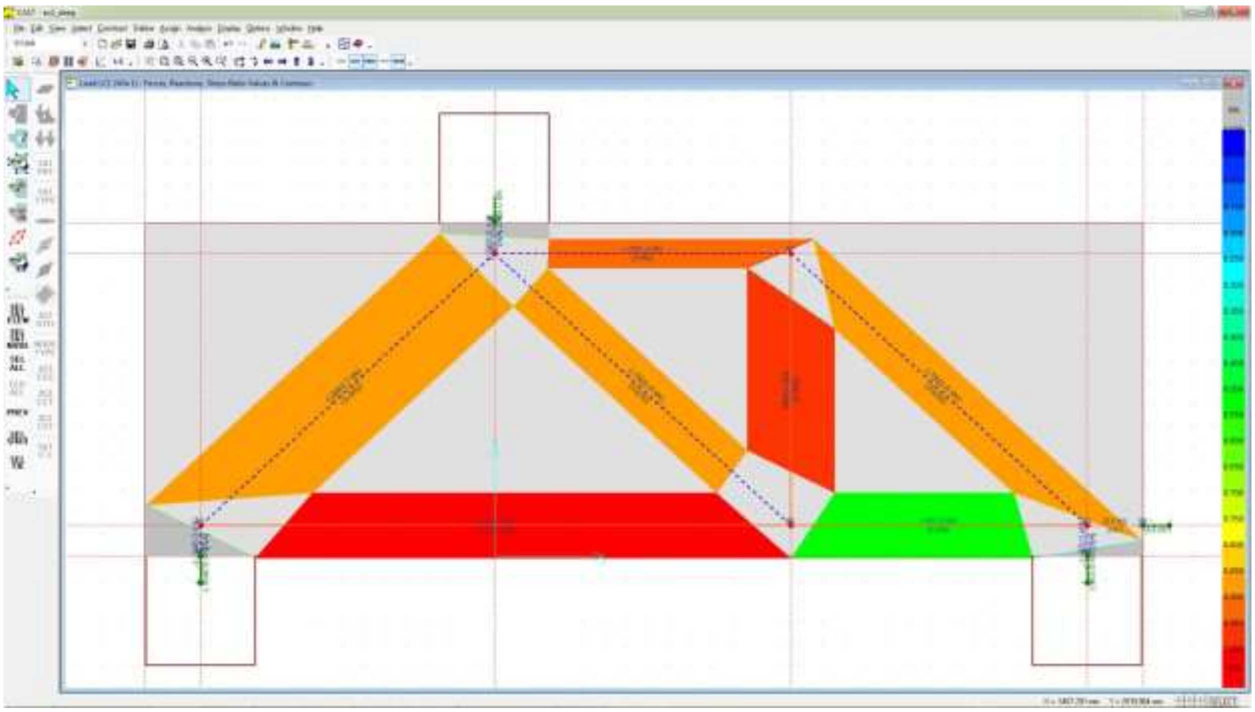


Figure 5.18 Analysis results

As shown by the analysis results, all elements are safe. A little overstressed in the horizontal tie can be noticed. Increasing the main rebar might be necessary.

Step 8: Check nodal zone stress

Stress at nodal zones must be checked to assure safety of the model. As shown in Fig. 5.19, stresses ratio at N1 for three perpendicular faces are confirmed to be less than 1.0

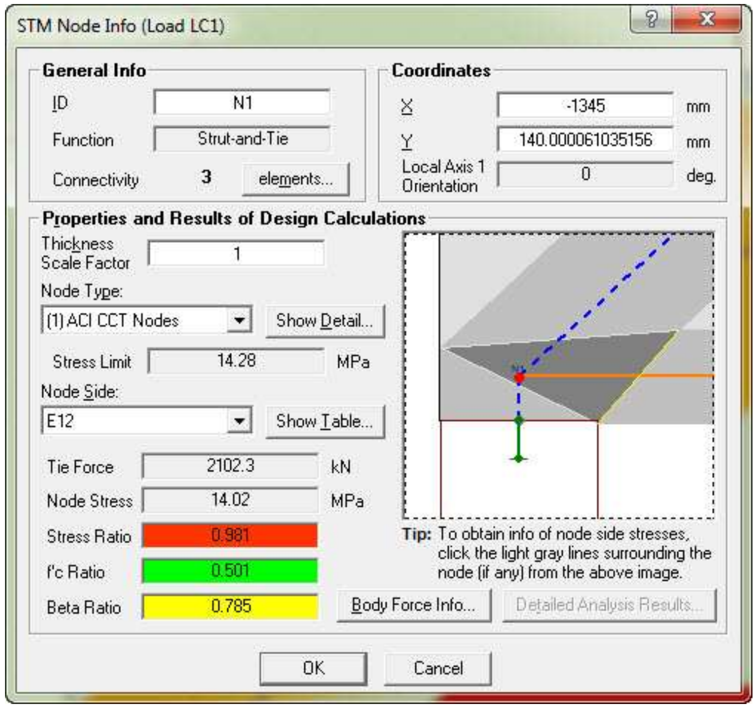


Figure 5.19 Stress check at nodal zone N1

As indicated by the results, the face perpendicular to the main tie is subjected to high stress ratio of 0.98. Rearrangement of rebar in tie may help to reduce the stress ratio, but it seems unnecessary for this example.

Step 9: Check for adequate development length. Since this problem is deep beam, minimum reinforcement according to ACI 10.6.7 skin reinforcement or ACI A3.3 must be confirmed. Moreover, maximum shear stress as stipulated in ACI Code 11.8.3 shall be checked.

5.3 Ledge Beam

This example shows an application of ledge beam. The 200 mm thick D-region is shown in Fig. 5.20. The ledge is subjected to vertical load 25 kN applied at bearing plate.

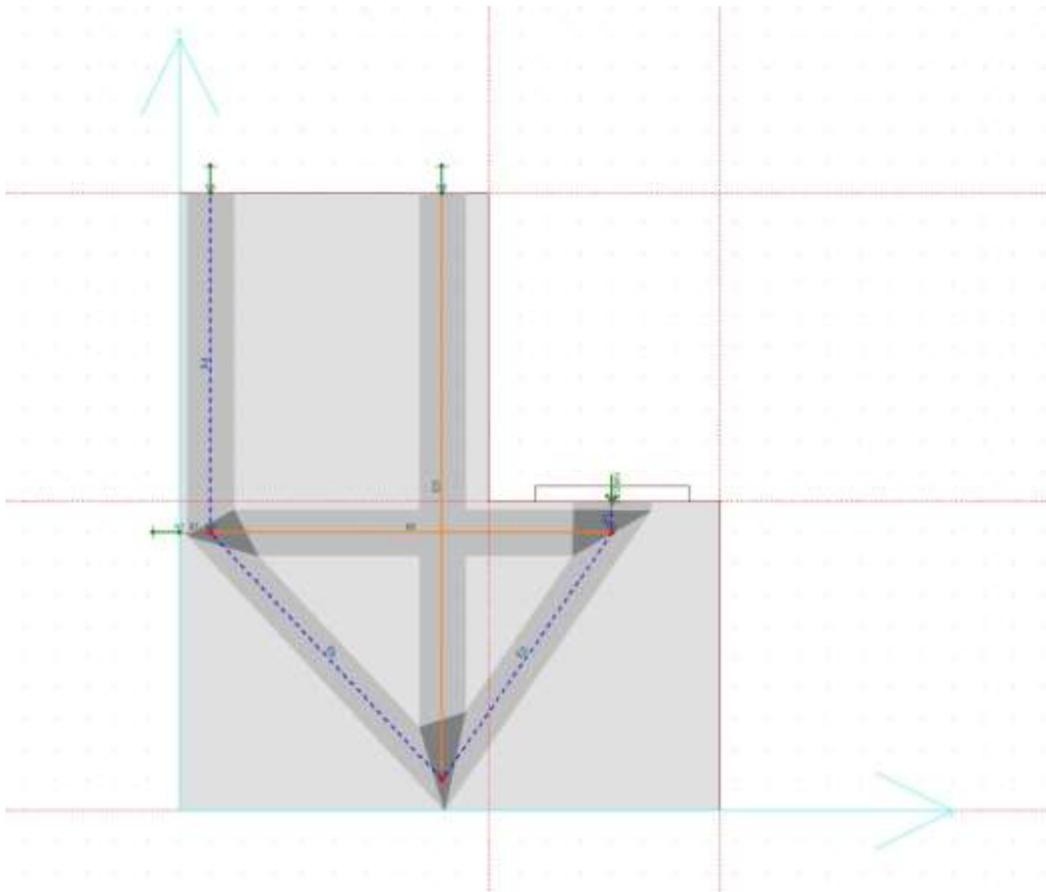


Figure 5.20 Outer boundary of D-region and strut-and-tie model

The model consists of two tie elements (E5, E6) and four compression struts (E1, E2, E3 and E4). A stabilizer (E7) is added to make the model stable. The analysis results are shown in Fig. 5.21. For a trial design, ties E5 and E6 are assigned as 2T12 (226 mm^2). The effective width of ties are 30 mm. Since lateral spreading of compression field is possible within the ledge, struts E2, E3 are assigned as bottle-shaped strut without reinforcement satisfying ACI318 A.3.3 (

$\beta_s = 0.60\lambda$). Compression field is confined within the beam web, so that prismatic strut ($\beta_s = 1.0$) is used for E4. The analysis result in Fig. 5.21 shows capacity ratio of the strut-and-tie model. Fig. 5.22 shows detailed calculation for strut E2. Investigation of nodal stress can be performed as similar to the previous examples.

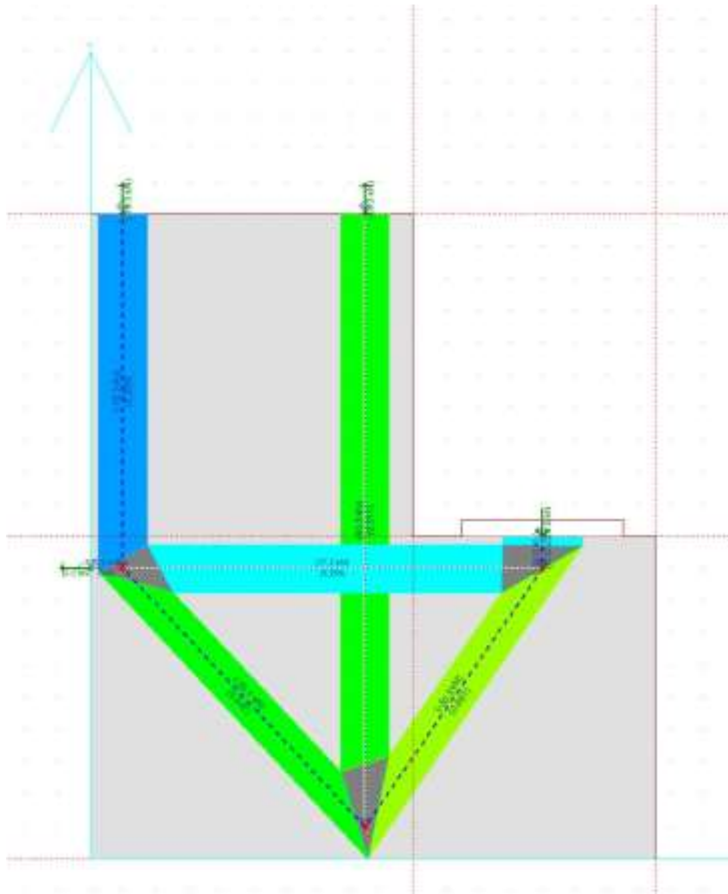


Figure 5.21 Analysis results

Figure 5.22 Detailed calculation for strut E2

References

KJ Bathe, Finite Element Procedures, Prentice-Hall, 1996

F Biondini, F Bontempi, PF Malerba, Optimization of strut-and-tie models in reinforced concrete structures, Proc. The Australian Conference on Structural Optimization, Sydney, Edited by GP Steven, OM Querin, H Guan, and YM Xie, pp. 115-122.

MP Collins, D Mitchell, Prestressed Concrete Structures, Prentice Hall, 1991

PW Cristensen, A Klarbring, An Introduction to Structural Optimization, Springer Verlag, 2008

AJ Dammika, N Anwar, Extraction of strut and tie model from 3D solid element mesh analysis, International Conference on Sustainable Built Environment (ICSBE-2010) Kandy, 13-14 December 2010

W Kanok-Nukulchai, & N Anwar, Space Truss Model for Design of Pile Caps. Asian Institute of Technology, 1996

X Huang, YM Xie, Evolutionary Topology Optimization of Continuum Structures: Methods and Applications, John Wiley & Sons, 2010

DA Kushma, TN Tjhin, CAST (Computer-Aided-Strut-and-Tie) Design Tools, ASCE Proceeding, doi: http://dx.doi.org/10.1061/40558(2001)142), 2001

QQ Liang, YM Xie, GP Steven, and LC Schmidt, Topology optimization of strut-and-Tie models in non-flexural reinforced concrete members, Int. Conference on Mechanics of Structures, Material and Systems, 17-19 February, 1999

JG MacGregor, Reinforced Concrete: Mechanics and Design, 3rd edition, Prentice-Hall, 1997

A Muttoni, J Schwartz, B Thuerlimann, Design of Concrete Structures with Stress Fields, Birkhauuser, 1997

AH Nilson, D Darwin, CW Dolan, Design of Concrete Structures, 14th edition, McGraw Hill, 2010

CG. Quintero-Febres, G Parra-Montesinos, and J. Wight, Strength of Struts in Deep Concrete Members Designed Using Strut-and-Tie Method, ACI Structural Journal, No. 103-S61, Technical paper, July-August 2006, pp. 577-586

J Schlaich, K Schaefer, and M. Jennewin, Toward a consistent design of structural concrete, PCI journal V.32, No. 2, pp. 72-150

A Vontobel, Calculating strut-and-tie models with Fachwerk and Fachwerk3D : Program documentation, June 2010

CK Wang, CG Salmon, Reinforced Concrete Design, 5th edition, Harper Collins, 1992

EL Wilson, Static and Dynamic Analysis of Structures, Computers and Structures Inc., 2010

JK Wight, Development of the Strut-and-Tie Method for Appendix A of the Building Code (ACI 318-08) http://www.concrete.org/tempComDocs/-19398/stm_-_sa_trip_2008__compatibility_mode_.pdf

JK Wight, GJ Para-Montesinos, Strut-and-Tie Model for Deep Beam Design, Concrete International, May 2003, pp. 63-70.

ACI 318M-08, Building Code Requirements for Structural Concrete (ACI318M-08) and Commentary, American Concrete Institute

Web resource

http://dankuchma.com/stm/CAST/

http://dankuchma.com/stm/CAST/tutorials/dcorbel/

http://fachwerk.sourceforge.net/index_en.html