ANALES de Edificación

Received: 21/10/2023
Accepted:12/11/2023

Anales de Edificación
Vol. 9, Nº3, 55-64 (2024)
ISSN: 2444-1309
DOI: 10.20868/ade.2023.5380

Método de las líneas de rotura a través del método de los elementos finitos. Aplicaciones con SAP2000

Method of break lines through the finite element method. Applications with SAP2000

José Javier Ferrán Gozálvez, Carlos Manuel Ferrer Gisbert, Miguel Redón Santafé, Esteban Gargallo Tatay, Juan Bautista Torregrosa Soler, Francisco Javier Sánchez Romero

Universitat Politècnica de València. D.I.R.A. Unidad de Construcción

Resumen-- **En la presente ponencia se expone como ha ido evolucionando el método de las líneas de rotura a lo largo de los años. Hoy en día con los ordenadores, se obtienen por contornos de colores resultados muy parecidas a las antiguas líneas de rotura, solo que se abre el campo de aplicaciones, especialmente en losas de hormigón armado. Para ello se ha utilizado el programa de elementos finitos SAP2000, con todo su aparato informático. Además, constituye la fundamentación teórico de una Tesis que los autores estamos desarrollando, pero en este caso con losas apoyadas sobre el terreno.**

Palabras clave— Método de las líneas de rotura; MEF; ejemplos SAP2000.

Abstract— **This paper shows how the method of break lines has evolved over the years. Nowadays, with computers, coloured contours can be used to obtain results very similar to the old break lines, but the field of applications has been opened, especially in reinforced concrete slabs. The SAP2000 finite element programme with all its computer equipment was used for this purpose. In addition, it constitutes the theoretical basis of a thesis that the authors are developing, but in this case with slabs supported on the ground.**

Index Terms— Breaking line method; FEM; SAP2000 examples.

I. INTRODUCTION

According to the Ultimate Limit State design method, the plastic calculation is a more suitable calculation procedure than the linear elastic calculation as it allows estimating the plastification or maximum stress that a section is able to withstand or the ultimate or collapse load that causes the ruin of the structure because it is transformed into a statically unstable mechanism or structure.

The plastic capacity of a section or structure may be limited by possible instabilities due to buckling of section elements, or geometric non-linearity (buckling), leading to the failure of the structure before the collapse mechanism is formed.

The standards base the calculation of a section or part on obtaining the exhaustion stress, which for many types of sections, when certain conditions are met, is considerably higher than the stress at the elastic limit.

The Ultimate Limit State (ELU) design in reinforced concrete is based on the depletion of the member, but not the design of the structure, which is elastic. In the case of reinforced slabs, it can be interesting to resort to these limit states of exhaustion, as is the case of plastification, by means of the calculation of the method of rupture or creep lines (Nilson et al., 1994; Kennedy et al., 2004; Park et al., 1987) and more recently (Tinoco et al., 2017; Gilbert et al., 2014; Stochino et al., 2019; Zhu et al., 2010) among others.

The calculation by means of the yield line method is an upper

JJ.F., C.F., M.R., J.B., and F.S. are associate professor at Agroalimentary and rural engineering department from Universidad Politècnica de Valencia. D. Ing. Rural y Agroalimentaria, Camino de Vera, s/n 46022 Valencia. E.G. is PhD student. at Agroalimentary and rural engineering department from Universidad Politècnica de Valencia. D. Ing. Rural y Agroalimentaria
Camino de Vera, s/n 46022 Valencia

limit state, i.e. its conclusions can lead to results higher than the real ones creating a collapse mechanism, but on the other hand it has been shown that for the most common cases the slabs resist more than those calculated with 1st order elastic methods.

The analysis of slabs by means of rupture lines was first proposed by Ingerslev (Ingerslev et al., 1923) and was disseminated by Honegstad (Hognestad et al., 1953).

It is not the aim of this paper to explain the method of breaklines (known since ancient times), which can be found in specialised texts. It is simply to explain how similar procedures are arrived at using the finite element method (in this case with SAP2000 v.21) (Wilson et al., 2009) and to try to find similarities and disagreements, the former being a method that acts on plastic lines and the latter a procedure that covers larger areas.

For this purpose, three exemplary cases will be introduced: a) the first one on a square slab with its four edges supported; b) the second one ditto but with the four edges embedded; c) and the last one the case of a slab with a concentrated load, whose development is the principle of the current methods for the dimensioning of rigid concrete slabs and pavements of which a Doctoral Thesis is being developed, described by Meyerhof (Meyerfof et al., 1962). The work we have developed covers 100 pages with a lot of casuistry, but due to limitations of space and content, we will limit ourselves to these three cases.

II. Experimental program

A. Elastoplastic calculation with SAP2000

Many advanced finite element analysis programs have a large gallery of elements capable of modelling elastoplastic behaviour. Plastic behaviour (definition of the stress-strain diagram) is implemented in the expressions defining the constitutive properties of the material. This behaviour is concentrated at the ends of the element, so that sufficient meshing is carried out to develop the plasticisation sections. From the definition and discretisation of the section, the program evaluates the necessary parameters for the calculation (for example, the moment/curvature curves for each axial value, highlighting the moment at the elastic and plastic limit, considering the limit deformations prescribed by the standards).

In SAP2000 (Wilson et al., 2009) the analysis can be simplified, so that with the definition of a few parameters, it tries to model the plasticisation process in a simple way, and thus a structure on which a conventional elastic analysis was performed can then be carried out, without many complications, a plastic analysis. It also uses automatic modelling tools or incorporates prescriptions or recommendations from the American standards.

It uses the concept of the "plastic hinge" or section that the designer considers to be susceptible to the formation of a hinge. In other words, the foreseeable necessary positions of plastic hinges are defined on a bar in such a way that their appearance and, if necessary, their development can be detected. The program only activates those that are plasticised, while the rest remain elastic.

The programme does not calculate the position of the spherical plain bearings. It is therefore necessary to review, for example in the case of bending spherical plain bearings, the moment diagram and check that in no section of a member does the acting bending moment exceed the plasticising moment. If this is not the case, it will be necessary to define a new hinge (i.e. formation of a possible hinge) at that point.

However, even if there is no section in which the deflection exceeds the plastification deflection, it is not possible to ensure that the positions of the plastic hinges are correct, due to stress redistribution.

The program has a wide variety of commands, simple or more sophisticated, to implement a plastic calculation in any structure. However, the main interest of the programme is the verification of the structural behaviour in seismic situations with the prescriptions or recommendations of the American standards. In steel, it refers specifically to document FEMA-356 (Federal Emergency Management Agency 5.5.2.2.2. Nonlinear Static Procedure) (FEMA, 2000).

The program internally performs the automatic calculation of hinges according to the FEMA-356 procedures, which simplifies the definition of the hinges quite a lot. In any case, it allows the user to modify the parameters. In this way they can be adapted to the requirements of your project and standards.

It offers various procedures that can be adjusted with different values of the calculation control parameters. The consequence is that it is advisable to carry out several analyses of the same problem to check the uniqueness, convergence, and realistic nature of the solution.

A given load acts on the structure. The program performs elastic calculations for each monotonically increasing fraction of this load or steps until, if possible, its full application. The sequence stops if a plastic hinge is activated and determines the load at that instant. From that point, increasing increments of the load are added again, and further spherical plain bearings can be activated, until the load acting on the structure is reached, or the calculation is stopped when the collapse mechanism is formed.

There are two methods for monotonically increasing load increments, which are discussed below.

B. Static Nonlinear Analysis

a) Load control

The program proceeds to perform a linear static calculation for each load level or step until successive plastic hinges are formed, imposing stress and strain boundary conditions on the hinges. The calculation becomes non-linear as the conditions of the structure change after each plastification.

The calculation terminates either when the defined load is reached or if the structure is unable to verify the plastification conditions (by transforming into a collapse mechanism or reaching the maximum allowable plastic deformations).

In this case, the program aborts the calculation (because the limits set for some of the control parameters are exceeded) or it enters a loop without achieving convergence (the program can be manually forced to interrupt and terminate the calculation, for which the program retains the results obtained up to that point).

The program calls pushover the previous analysis supported by the automatic generation of plastic hinges.

This is the simplest and most basic calculation method, but it can present problems of convergence to a solution or give incorrect results in certain cases of geometric instability. The number of steps can be small (between 20 and 50). It is advisable to check that no changes occur as the number of steps increases, which happens in many cases, ensuring that the calculation is adequate.

It can be activated in the programme in two ways:

- Static Nonlinear analysis, which is the usual procedure (Static Nonlinear).
- Static Nonlinear Staged Construction (Static Nonlinear Staged Construction). In this case the program calculates for the defined steps, not adjusting the load value to the occurrence of hinges. This analysis is of interest in singular cases where it is desired to define events (removal or addition of members, change of section, occurrence of joints, partial loads, etc.).

b) displacement Control

A variant of the non-linear static analysis is the one that aims at reaching a given deformation at a given point, target displacement or displacement control, instead of reaching the acting load. The type, value, and direction of the movement to be monitored must be selected. For example, in a gable portal frame, the vertical displacement of the ridge node. The behaviour of the structure must be known, and previous analysis must have been carried out.

The programme will try to achieve this target displacement or will be interrupted in case the structure collapses or there are calculation problems. Not to be confused with the imposed displacement type of loading assumption. It can give good results in P-Delta problems with large displacements.

This method can provide a solution when load control is not achieved. This is because the program can perform load decrements to overcome, if necessary, transient imbalances. It is an alternative to non-linear dynamic analysis, much simpler to activate and requiring less computational time.

c) nonlinear time-history análisis

Generally used to determine the stepwise dynamic response of a structure subjected to any time-dependent loading. It considers the inertial mass, damping and stiffness of the structure. It can include non-linear material behaviour (plastic hinges) and geometric non-linearity (P-Delta effect and large displacements).

In general, the calculation dealt with in this topic is of a static nature. The fact that a dynamic calculation is carried out is because it is a more robust calculation method with a greater capacity to converge to a solution when non-linearities are present, especially in the case of large displacements, rather than trying to simulate dynamic behaviour, although this better approximation to reality is also achieved if the dynamic calculation parameters are appropriate.

This type of calculation involves several dynamic variables that must be defined for this type of discrete and slowly growing loading problem:

- Inertial masses cause inertia forces which dissipate in time through damping, but which can influence the calculation results. In general, it is sufficient to activate the self-weight of the structure or to activate the permanent loads as inertia masses.
- Inertial masses cause inertia forces which dissipate over time through damping, but which can influence the design results. In general, it is sufficient to activate the own weight of the structure or to activate the permanent loads as inertial masses.
- Load application time: This should be long (several minutes) to ensure very slow growth. The load can even be split up, so that up to a certain load value the analysis is static, to continue the dynamic analysis during the development of the spherical plain bearings up to the collapse.

If the time were fast (a few seconds), the calculation would be altered when it appears:

- Large inertia forces
- High transients or vibrations

In both cases, these effects can be reduced with a prolonged load application time or by keeping the applied load constant for a time interval when it reaches its maximum value so that the accumulated energy is dissipated. Two parameters influence this energy dissipation which could have a major influence on the calculation that can only be known by testing:

- Damping: two types of damping (Rayleigh) can be activated, one proportional to the mass matrix (in the manner of movement within a viscous fluid) or proportional to the stiffness matrix (that of the structure itself). The former acts on high period modes of vibration and the latter on low period modes of vibration.

A value of 0.02 damping coefficient is usually adopted for steel structures and should generally be increased for our purposes. If it approaches the supercritical value, transient vibrations do not occur, but inertia forces increase. A value between 0.05 to 0.15 can be tested at first for both types of damping and reduce the value proportional to the stiffness matrix by increasing the load application time.

- Hysteresis: the way in which a spherical plain bearing responds to the loading-unloading-reloading cycle:
- Isotropic: in each cycle the compressive and tensile strengths are increased, and the stiffness is maintained. This is the maximum energy dissipation. Transient vibrations cause an increase in the resistance of the hinges (they support a greater moment), which can lead to errors as the load capacity increases.
- Kinematic: considers the Bauschinger effect in metals, which consists in the fact that when a metal is deformed and exceeds the elastic limit in one direction, when it is deformed in the opposite direction, the limit of

proportionality is lower. Therefore, the isotropic strength increase does not occur in the elasto-plastic section of the stress-strain diagram of steel, on which we are working.

- Takeda and Pivot for reinforced concrete

If the load is applied very slowly or the damping is high, this parameter has no influence on the calculation.

The calculation is based on the load application time:

- The defined time step size is increased each time until the load application time is reached. These steps have the same meaning as in the non-linear static analysis. It is possible to set between 1000 to 10000 steps of 1 second duration.
- A function (time history function) is defined which relates the load and time, generally linear, so that for each instant a fraction of the load corresponds to it, in this case, proportionally. This type of load is called pulsating (transient or RAMPTH or triangular pulse that grows linearly from zero to the maximum value of the load). Such a load can be held fixed for an additional period so that transients (vibrations) arising from the dynamic nature of the load are reduced by damping. The aim is to simulate a quasi-static application of the load.

In a conventional dynamic analysis, this function represents the real variation of the load as a function of the load application time.

Therefore, each time step size is a fraction of the applied load, which is increased in each calculation until it is reached.

On the other hand, in each calculation performed for each fraction of time, the equation of motion of the structure is solved, which in turn is an iterative calculation, and which may include existing non-linearities (plasticity with hinges and P-Delta effect). The non-linear dynamic calculation is defined by the method of direct integration.

The number of variables involved in the calculation increases. Testing must be carried out to ensure that changes in their values do not cause significant changes in the solution:

It is important to set the Time Step Data appropriately. A high Number of Output Time Steps and a low Output Time Step Size are required to obtain adequate accuracy and convergence. The product of the two is the load application time, which is usually assigned a unit of time (e.g. 1000 steps of one second is: 1000x1 = 1000 seconds) during which the entire load is applied (Rampth function). A high value may increase execution times if no changes occur during execution. If it is necessary to keep the load fixed, a suitable value must be set. It can range from 1.5 to 4 times the initial value. The calculation times increase in similar proportion.

For direct integration, the HHT (Hilber-Hughes-Taylor-Alpha) method is often suggested as the most recommendable. If $\alpha = 0$ it is equal to the Newmark method with $\beta = 0.5$ and $\beta = 0.25$. With $\alpha = 0$ the highest accuracy of the available methods is obtained ($0 \leq \alpha \leq -1/3$). This method introduces some damping in the structure.

The program adjusts to the defined time steps, so that the results refer to each step, not to the achievement of a new ball joint. This does not usually present a problem of accuracy because, as mentioned above, the time steps are defined very small.

The calculation requires more time and storage capacity. A larger number of calculation parameters (masses, damping, load/time function, etc.) and calculation control parameters (number of iterations, tolerances, etc.) must be controlled.

Its advantage is that after fixing the parameters, the solution is unique, with a better or worse approximation when varying them. On the other hand, the static method can offer different solutions when the parameters are modified. It is a more realistic behaviour (even if the load application time function is not real).

It is often necessary if geometrical non-linearities need to be activated, such as the fire design of some structural types (gable portal frames).

d) Definition of plastic hinges

The basis of the programme's plastic calculation is to fix possible plasticisations or hinges in the sections where they are likely to form. In this way the programme activates the plastic properties if the forces in the section containing a hinge reach the plasticising forces.

We will use the term plastic hinges for these possible plastifications of whatever nature (stress), although literally a hinge is due to the action of a bending force. These hinges can be:

- Due to any type of stress (N, My, Mz, Vy, Vz, T) or combination of stresses.
- They may be in different sections or in the same section, in which case:
- They can be uncoupled stresses (independent or non-interacting).
- They can be coupled forces (related to an interaction curve), which is the usual case.

The program does not calculate either the position or the length of plastic spherical plain bearings. In general, hinges are usually formed at bar ends, at points of application of point loads or at changes of cross-section. The problem arises in uniformly loaded beams, where the position of a possible hinge inside the member, caused by a maximum relative bending, is unknown. To solve it, many possible hinges can be generated in that member (10 or 20), or else proceed by trial and error (place the hinge at one point and after the calculation change its position or add another one).

According to the static method, the different load factors obtained in each trial and error (meeting the equilibrium conditions) have a factor lower or equal to the actual collapse load factor, so they are on the safe side as long as in all sections $M_{Ed} \leq M_{pl}$. Therefore, the highest load factor will be taken.

The hinge type for plastification due to the interaction between axial (P), bending (M3) and bending (M2) is highlighted. It is the PMM hinge. They constitute a volume bounded by concave surfaces of limit resistance in each of the octants delimited by the limit stress trihedron. A point inside the volume signifies a combination of forces that does not exhaust the section. In the plane it is a concave curve in each quadrant, defined by the equations according to the standard used.

The programme has more advanced possibilities than the classical plastic calculation, such that in some aspects it can approach a non-linear elasto-plastic analysis:

- The non-linear response of the steel in the moment-curvature diagrams of the defined sections is considered, with the eventual consideration of a relevant axial.

- In the calculation, the plastic hinge affects a specific cross-section, so that the plastic deformation is only accounted for at that point. The program allows you to define the length of the hinge where the integration of the curvature will be performed. It is also possible to define hinges close to a distance from each ball joint (Frame Hinge Overwrite), which is also interesting when the precise position of the hinge is unknown, or when a more progressive plasticisation is desired if the length of the ball joint includes two or more hinges.

- The calculation is carried out with discretely increasing fractions of the load introduced (except for load levels at which a hinge is formed). If the load is sufficiently high, the ultimate strength of the sections can be reached and, if necessary, the structure can even be transformed into a mechanism.

For this purpose, the program performs, step by step, a linear elastic calculation for each load level or state (depending on the load if the calculation is static or depending on the time if it is dynamic), until, if the loads are sufficiently high, at some hinge, the stresses reach the elastic limit fyd of the steel, thus developing plastic deformations in the section involved. At that section, elastic deformations will no longer be considered.

- At each step or load level, the programme can include a P-Delta calculation (analysis that considers the secondary forces and stiffness modification in the structure due to the deformation of the structure), p-delta if each member has been meshed and even P-Delta with large displacements (equilibrium with the forces on the deformed position).

These calculations are mandatory if the structure is translational. The important consequence is that the plastic behaviour of translational structures can be analysed by considering the 2nd order theory.

- The effects of axial and/or shear forces with respect to bending can be considered when their influence on plasticization and hinge formation is relevant. This is the axial-bending-shear interaction.

- Elastic instability of the members (buckling) can be considered. The procedure is to use the definition of the P-M3 interaction for this purpose. For this, in tension it is replaced in the expression Npl,Rd, while in compression by the buckling resistance Nb,Rd (buckling factorc method).

If carried out in conjunction with a P-Delta analysis, conservatively, the buckling beta can be taken as unity. Between the two planes, the plane that provides the greatest slenderness is taken. If a local and global P-Delta calculation with geometric imperfections is performed, the calculation includes instabilities.

- The programme also analyses the residual stresses and strains if unloaded, and the type of hysteresis cycle (how the stress-strain diagram is traversed in load/unload cycles, or how energy is dissipated in those cycles) can be selected. It is of interest in structures with dynamic or repetitive loading without relevant fatigue (earthquake). Describe the different devices used in the development of the work.

III. Results and discussion

A. *Edge and corner behaviour square slab*

a) Edges

At a free or simply supported edge, bending and torsional deflection are zero. On the fracture line there are principal bending forces (maximum and zero without torsion), so the fracture line must be perpendicular to the edge.

For this reason, it has been proven that when a fracture line intersects an edge with a certain inclination □, it changes direction in the vicinity of the edge to reach the edge more perpendicularly. This is due to the shear forces (from the support) in this area, which generate torsional forces that cause this change in direction. As two equal forces of opposite sign generate this deflection, their virtual work is zero. Therefore, it does not affect the breaking load, only its configuration. It occurs at a very short distance from the edge.

b) Corners

In the cases analysed it has been assumed that in the corners the breaking lines intersect at the corner apex. However, the break line bifurcates before reaching the corner. Two cases arise:

- The support only acts in one direction, allowing the slab to rise in the corner by rotating with respect to the line a, b in Fig. 1, i.e. the support does not act if the reaction changes sign and therefore does not restrict the upward movement. There are two positive break lines ac, bc bifurcating from the diagonal break line. There should be reinforcement on the underside in that area in both directions.
- The support restrains movement in both directions, preventing the slab from uplift at the corner.

The shear stress at break that appears at the corner and that must be supported to prevent the corner from separating from the corner is:

$$V = m \cdot (\cot \alpha + \cot \beta) \qquad Para\ \alpha = \beta = 45^{o}: \quad V = 2 \cdot m$$

where α and β are the angles of the line of rupture with the sides of the corner.

There must be reinforcement or reinforcement to negatives (in the upper face) which, if it is not sufficient, is the cause of

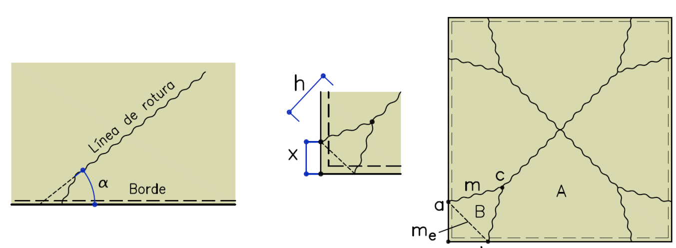

Fig. 1. Breaking lines on an edge and in a corner

the appearance of a third line of rupture that coincides with the axis of rotation ab. As we increase the negative reinforcement, the line a,b moves parallel to the corner. With sufficient reinforcement, the line ab disappears, and the break shape is simply the diagonal. It is envisaged to provide such reinforcement according to ACI 318-56, giving the corner sufficient strength.

c) Assessment

This break configuration may be more unfavourable in some cases. The problem is that the analysis becomes analytically complicated.

For example, in a square slab of side a subjected to a uniform load in which the upper reinforcement in the corner (line a,b negative), the least stressed area, is usually less or equal to the lower reinforcement m (line ac and bc, positive), two additional geometric variables appear, which in Fig. 1 are the position of the bifurcation point x and that of the intersection point with side h. We denote the ratio k $k = m_e/m$.

The length h is independent of x $h = \sqrt{6m \cdot (1+k)/q_u}$

From Table 1 if the reinforcement on both sides is the same at the corner, there is no bifurcation point. For k = 0.5 the effect is small and for k = 0 with no negative reinforcement at the corner, the bifurcation is maximum $0.5 \cdot x + h/\sqrt{2} = 0.45 \cdot a$ and q_u is reduced 22-100/24 = 9%. For other types of slabs, the more the corner angle differs from 90o, the more the ultimate load can be reduced by up to 35%.

TABLE I
POSITION OF THE BIFURCATION POINT AND ULTIMATE LOAD FOR VARIOUS RATIOS BETWEEN THE CORNER REINFORCEMENT AND THE REST OF THE SLAB. (*) COINCIDES WITH THE DIAGONAL.

k	x	h	m/q_u
0	0.159·a	0.523·a	$a^2/22$
0.25	0.110·a	0.571·a	$a^2/23$
0.5	0.069·a	0.619·a	$a^2/23.6$
1	0	0.707·a*	$a^2/24$

B. Concentrated loads

If a point load acts on an interior point at a certain distance from the supports (Fig. 2), a negative break line of circular-polygonal shape is formed at a certain distance from this point and positive break lines in radial direction starting from this point to the circular break line. Theoretically the number of radii is infinite and the surface tends to be conical.

The reinforcement is orthogonal in both directions. The plastic moment per unit length at positive bending or bottom face is m_r which acts according to the radius of length r and radius spacing an angle α. The plastic moment at negative bending or upper face is mc which acts according to the chord and has length r·α.

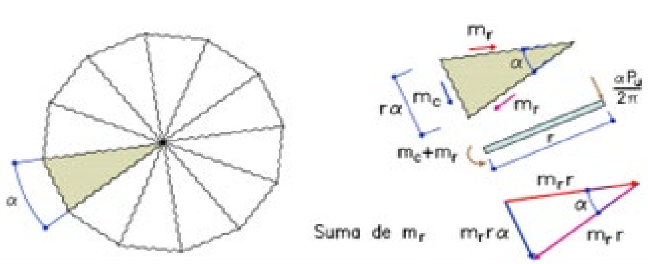

Fig. 2. Break lines of a slab subjected to a point load

Fig. 2 shows the moment equilibrium on one of the slab fragments or sectors. The vector sum of the moments according to the radii $m_r \cdot r$ es $m_r \cdot r \cdot \alpha$.. This moment is added to the total negative moment $m_c \cdot r \cdot \alpha$, both having the same direction. On the other hand, the fraction of the load P_u on the sector α/2π originates the moment that balances the sector $\alpha P_u r/2\pi$. The value of P_u is obtained:

$$(m_r + m_c) \cdot r \cdot \alpha - \frac{\alpha \cdot P_u \cdot r}{2\pi} \quad \rightarrow \quad P_u = 2\pi \cdot (m_r + m_c)$$

The ultimate load P_u does not depend on the radius r nor on the number of sectors or angle α, which means that any failure configuration is possible without influencing P_u. Furthermore, it is also dependent on the shape of the slab and its edge conditions, so that this could be its failure typology depending on the case. The ultimate load P_u only depends on the plastification moments.

It is more realistic to consider the point load distributed over a small circular area. The ultimate load P_u decreases with increasing ϕ. If the slab is embedded at its edges, the fracture circle is tangent to the embedded edges.

C. Examples

The configuration of the mechanism and the ultimate load of several simple cases are to be determined. The load q is uniformly distributed over the entire slab. In some examples the slab is isotropically reinforced, i.e. equally reinforced in both directions. The embedment deflection is m for any angle of inclination of the line of rupture.

a) Balance method

The collapse mechanism is known thanks to the double symmetry. The lines of rupture are the diagonals and are all positive (limahoyas) (Fig. 3). On the supported side of one of the fragments, we perform the equilibrium of moments.

The moment caused by the qu-load acting on A/4 whose resultant is at a/6 with respect to the supported edge is in

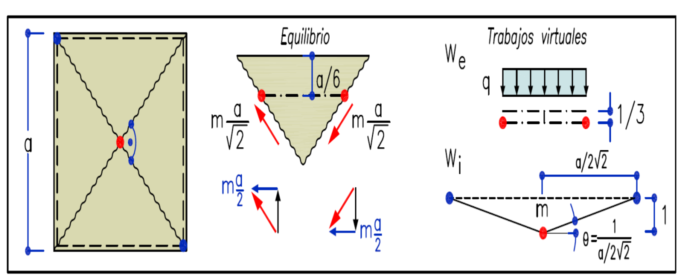

Fig. 3. Analysis of a square slab with hinged supports by the equilibrium method and by the virtual works method

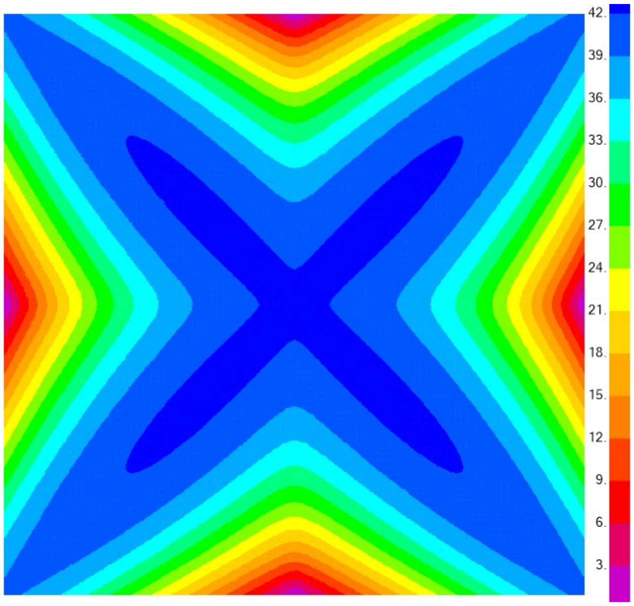

Fig. 4. Result with the finite element method (view)

equilibrium with the side-parallel component of the plasticising moment along the two lines of rupture:

$$\left(q_u \cdot a \cdot \frac{a}{2} \cdot \frac{1}{2}\right) \cdot \left(\frac{a}{6}\right) - 2 \cdot \left(\left[m \cdot \frac{a}{\sqrt{2}}\right] \cdot \frac{1}{\sqrt{2}}\right) = 0 \quad \rightarrow \quad q_u = \frac{24 \cdot m}{a^2}$$

b) Virtual work method

In the centre of the square slab we assume a unit displacement Δ = 1. The external work caused by the uniform load q_u is equal to the internal work due to the plastic moment m at the lines of rupture.

The external work is 4 times that of the fragment of area A/4 of average displacement 1/3.

$$W_e = 4 \cdot \int_{A/4} q_u \cdot \Delta \cdot dA = 4 \cdot \left(q_u \cdot a \cdot \frac{a}{2} \cdot \frac{1}{2}\right) \cdot \left(\frac{1}{3}\right) = \frac{q_u \cdot a^2}{3}$$

The constant twist of any line perpendicular to a break line is obtained from the diagonal through the central point down the unit. The work is the plasticising moment by the twist along the entire line of rupture:

$$W_i = 4 \cdot \left(m \cdot \frac{a}{2\sqrt{2}}\right) \cdot \left(\frac{2}{a/2\sqrt{2}}\right) = 8 \cdot m$$

$$W_e = W_i \quad \rightarrow \quad q_u = \frac{24 \cdot m}{a^2}$$

An elastic calculation gives us a maximum moment at the centre of the square of value:

$$m = 0.042 \cdot q_{elas.} \cdot a^2 = q_{elas.} \cdot a^2/23.8$$

$$\frac{q_u}{q_{elas.}} = \frac{24}{23.8} = 1.01$$

In practice, the same limit load is obtained in elasticity and rupture.

It can be seen in the SAP2000 calculation for μ = 0.15, unit load and unit sides that a fairly fixed maximum deflection has been obtained in the central diagonals: principal deflection Mmax = 0.042.

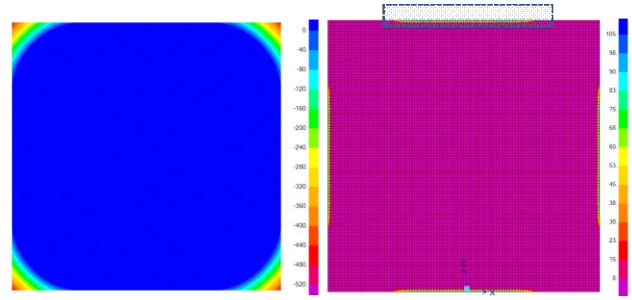
Fig. 5. Corner view. MEF

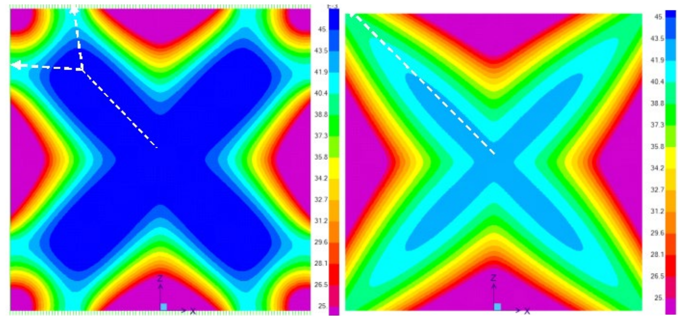
Fig. 6: FEM results including corner bifurcation

As soon as the load increases slightly, rupture lines form on the diagonals of the square and the slab collapses. The plastic reserve is practically zero, similar to the case of a simply supported beam.

c) Bifurcation of break lines at corners

In Section 6.7 the splitting of the break lines at the corners instead of intersecting with the corner was briefly explained.

The elastic calculation with SAP2000 reflects this onset of behaviour in a slab whose edges or sides only support positive (compressive) reactions, which causes the corners to rise, i.e. have positive displacements.

This is reflected in the deformation diagram on the left, where only the colour has been activated at the nodes with upward displacement, and in the pressure diagram at the support on the right.

The curved shape of the boundary of the change of sign of displacements will transform into a straight line of zero displacements as the plasticisation progresses. The first positive reaction occurs 0.23 m from the apex, almost ¼ of the side length.

The Mmax moment diagrams are shown below: on the left with corners that can be lifted and on the right with sides fixed in both directions.

It can be seen how the beginning of the bifurcation appears, which does not occur when the supports act in both directions. The colour scale being the same in both drawings, it can be seen that the one with edges that can be lifted has higher moments because it lacks the half bending (torsional stiffness) that the corners can provide.

The deflection which was 0.0424 increases to 0.0469 due to this circumstance. This is 10% which is quite close to the 9% decrease in plastic capacity (qua2/24 becomes qua2/22) quoted in Nilson and Winter, Design of Concrete Structures (Nilson et al., 1994).

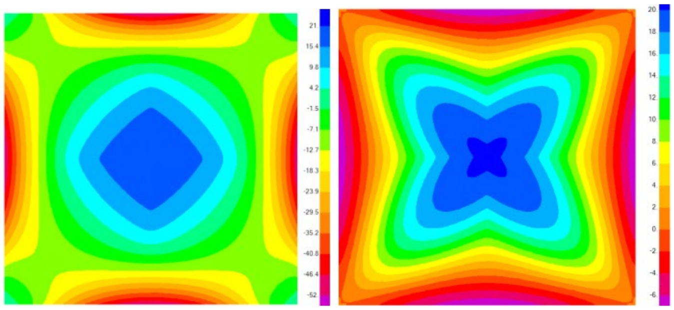

Fig. 7. View results with SAP2000

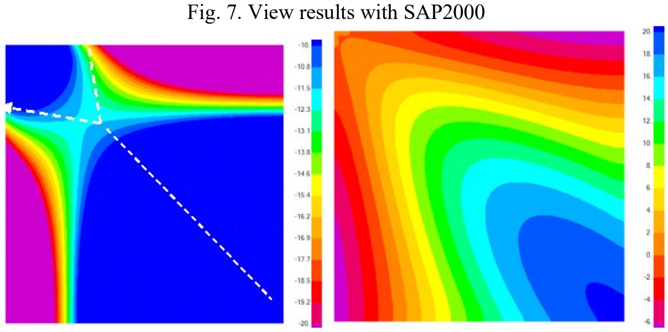

Fig. 8: Corner view with SAP2000

D. *Square plate with recessed edges*

a) Virtual work method

Four additional negative break lines appear at the recessed edges. The external work is the same as that assessed for the supported plate. To the internal work is added those due to plasticisation of the recessed edges.

The constant twist of any line perpendicular to a line of rupture is obtained from the diagonal through the centre point down the unit. The work is the plasticising moment due to the twist along the entire line of rupture:

$$W_i = 4 \cdot \left(m \cdot \frac{a}{2\sqrt{2}}\right) \cdot \left(\frac{2}{a/2\sqrt{2}}\right) + 4 \cdot (m \cdot a) \cdot \left(\frac{1}{a/2}\right) = 12 \cdot m$$

$$W_e = W_i \quad \rightarrow \quad q_u = \frac{36 \cdot m}{a^2}$$

By means of an elastic calculation we obtain the maximum moment at the centre of the square and the maximum negative moment at the centre of any recessed side:

$$m_{(-)} = 0.052 \cdot q \cdot a^2 = q_{elas.} \cdot a^2/19.23$$

$$m_{(+)} = 0.021 \cdot q \cdot a^2 = q_{elas.} \cdot a^2/47.62$$

$$\frac{q_u}{q_{elas.}} = \frac{36}{19.23} = 1.87$$

The large plastic reserve achieved with the recessed sides can be seen.

The moment diagrams Mmin (maximum at the centre of the sides) and Mmax (maximum at the centre of the square) are shown.

In the SAP2000 calculation, a maximum negative deflection is obtained at the centre of the sides: principal deflection Mmin = 0.051. The positive deflection is maximum at the centre: Mmin = 0.020. Both are very similar to those obtained from the tables.

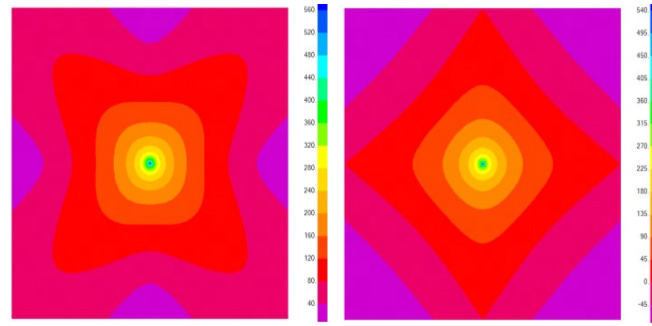

Fig. 9. Maximum moment view (left). Minimum moment (right)

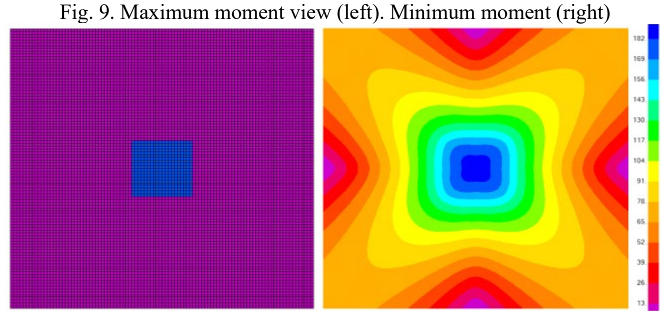

Fig. 10: Resulting view with square plate

The ratio between the two moments is 51/20 ≈ 2.5, so the positive bending moment still has plenty of room for growth until the plasticising moment is reached.

It can be seen in the Figures (¼ of the slab is represented) that at the corners the bifurcation of rupture lines can be generated even if the support acts in both directions. This is due to the fact that the moment Mmin (left) is higher at a certain distance from the corner and the positive Mmax (right) decreases until it cancels out at the corner (as opposed to the supported one).

E. *Square plate supported with point load in the centre*

Square plate supported on its 1 m side contour under a load at its centre. The reinforcement on both sides is identical so that the plastic resistance to positive mr and to negative mc are equal (m).

For a wreathed crack shape as shown in Fig. 4, the ultimate load Pu depends on the radius of application of the load:

$$Pu = 4\pi \cdot m = 12.56 \cdot m$$

But if a diagonal configuration is adopted, the ultimate load is Pu = 8-m, obtained by the virtual works method.

a) SAP2000

The positive moment diagram Mmax (left) and the negative moment diagram Mmin (right) are attached. The maximum bending moment Mmax is m = 0.575-qelas. The lines of rupture will be radial from the point of load application.

It can be seen that at the corners the maximum deflections are negative, and a negative moment diagram has been included for more detail. As the load is unit load: m. = 0.575-$P_{elás.}$, so $P_{elás.}$ = 1.739-m.

Mmax can presuppose the positive radial breaklines. Mmin could be the origin of the negative circular rupture line if this configuration prevails over the diagonal one.

It can be seen how the circles of Mmax close to the point load are transformed into squares with rounded corners, which are approaching the corners of the slabs. The higher absolute value towards the corners may anticipate that the diagonal rupture line will prevail over the radial one as the load increases, as plastification increases from the centre of the slab.

The ratio of the ultimate load to the elastic load is:

$$P_u/P_{elas.} = 8/1.739 = 4.60$$

There is a large difference that may be due to the large bending gradient between the sides and the centre of the slab. It is necessary to remember that for this load to be reached, in the elastic domain the reinforcement of the slab would have to be sized to resist such a high moment. In practice, this moment is distributed over a certain width (redistribution of moments) and can be reduced by half or a third.

In the following example the unit load is distributed over a square of 0.2x0.2 sides. As the load is distributed, the moments are also distributed and the maximum moments decrease.

The maximum deflection Mmax is m = 0.188 P_u, P_{elas} = 5.319.

Fig. 10 is similar to that obtained with point loading, but the moment gradient is much smaller.

The central square of 20x20 cm is attached. It can be seen that the average moment in this area is not as different as suggested by the maximum moment (0.575) with the point load, which could be around 0.23 (2.5 times smaller) while with the distributed load it would be 0.175 much closer to the maximum 0.188.

The ratio of the ultimate load to the elastic load is:

$$P_u/P_{elas}. = 6.63/5.319 = 1.25$$

This value is more realistic than that obtained as a point load because the concentrated load will be spread over a more or less small area.

IV. CONCLUSIONS

A. Breaking lines with SAP2000

The SAP2000 program has instructions that allow the definition of a plastic behaviour of a reinforced concrete section. The procedure has in common with bar structures the establishment of load increments until the structure can no longer balance them, thus obtaining the ultimate, collapse or ultimate load. However, in the case of reinforced concrete, there are differences in the methodology:

- The ball-and-socket joints are not defined as they are for bar structures. The deformation and moment diagrams must be interpreted to know the rupture configuration.
- It is not necessary to know a priori the location of the yield or rupture lines. Nor is it necessary to know the plasticisation length, as they will be generated as the loads increase. The graphical representation will allow us to know the onset of plastic behaviour when it is no longer linear.
- In steel, the plastification moment Mp is easy to evaluate in any type of section. In reinforced concrete, the behaviour of the section must be simulated from the reinforcement, the mechanical properties of the steel and the concrete, to obtain an ultimate moment Mu in a similar way to the parabola-rectangle method, for example.
- The behaviour will not be modelled as rigid-plastic (elastic deformations are neglected) but elasto-plastic. It will be seen that when modelling as rigid-plastic (high E) the results are not correct. As the plate fragments are not flat surfaces, the break lines are not straight.
- In SAP2000 the elements will be defined as Shells because when defining them as Plate to eliminate membrane forces, errors arise during the calculation.

Therefore axial forces F11, F22 or membrane forces may appear. If the deformations are small, so are the secondary membrane forces (axial forces) during the formation of the collapse mechanism. Although resisting by axial forces can provide additional load capacity by transforming the bending plate into an axially loaded plate, as in bending the tensile reinforcement reaches fyd, and the redistribution with the compressed reinforcement is small, the membrane effect will be small. It was found that by activating the P-Delta Large Displacements calculation, the resisted load is very similar.

REFERENCES

FEMA 356, Prestandard and commentary for the seismic rehabilitation of buildings, Federal Emergency Management Agency, Washington D.C., November 2000

M. Gilbert, L. He, C.C. Smith, C.V. Le, automatic yield line analysis of slabs using discontinuity layout optimization, DOI: 10.1098/rspa.2014.0071, Proceedings of the Royal Society, 2014

E. Hognestad, yield line theory for the ultimate flexural strength of reinforced concrete slabs, vol 24, nº 7, ACI Journal, pp. 637-656, 1953

A. Ingerslev, the strength of rectangular slabs, vol1, nº1, journal Instit. Struct. Enginners, p.p. 3-14, 1923

G. Kennedy, C.H. Goodchild, Practical Yield line design, Concrete Centre, Surrey, 2004

G.G. Meyerfof, Load carrying capacity of concrete pavements, Journal of the Soil Mechanics and Foundation Division, Proceedings of the American Society of Civil Engineers, June 1962.

A.H, Nilson, G. Winter Diseño de estructuras de concreto, úndecima edición, capítulo 14º Analisis de losas mediante líneas de fluencia ed., Mc Graw Hill, Santafé de Bogotá (Colombia) , 1994, p.p. 435- 456.

R. Park, W.L. Gamble, Losas de concreto reforzado, Base de la teoría de la línea de fluencia, capítulo 7º, Editorial Limusa, 1987 p.p. 315-377.

F. Stochino, F. López Gayarre, Reinforced Concrete Slab Optimization with Simulated Annealing, DOI: 10.3390/app9153161, Applied Sciences, MPDI, 2019

H.A. Tinoco, numerical limit analysis of reinforced concrete slabs using a dual approach and conic programming, Structural Multidisc Optim, 2017, DOI10.1007/s00158-016-1582-3, Springer, 2017

Wilson E.L. Integrated structural analysis and design software SAP2000. Computers and Structures, Inc. Berkeley, California. http://www.csiamerica.com , 2009 (continuously accessed)

Y. Zhu, Y.X. Zhang, Nonlinear finite element analyses of FRP-reinforced concrete slabs using a new layered composite plate element, DOI 10.1007/s00466-010-0485-1, Computer Mechanics, Springer, 2010