Concrete One-way Slabs

By using the [Concrete Member] design module linked to the [Analysis] design module, one-way slabs can be designed. The following tutorial will outline the procedure for a few common one-way slab arrangements.

**Simply supported One-way Slab**

The first example is a simply supported concrete slab spanning 4.8m, supporting a superimposed dead load (finishes) of 0.5kPa and a domestic live load of 1.5kPa.

An initial trial thickness of 190mm will be used with the bottom reinforcement being ductile N12 at 200mm centres, and top reinforcement being low ductility SL82 mesh. Top cover will be 20mm, and bottom cover will be 30mm (Refer below). A strength of 32MPa is used for this example.

Using the above configuration, the design parameters are input into [Concrete Member] as shown below.

### Geometry

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete strength (f’c)</td>
<td>32 MPa</td>
</tr>
<tr>
<td>Depth (D)</td>
<td>190 mm</td>
</tr>
<tr>
<td>Web width (Wb)</td>
<td>5 mm, (Slab)</td>
</tr>
<tr>
<td>Slab type</td>
<td>Q (One-way, Two-way &amp; Comb, Two-way &amp; Walls, Two-way &amp; Footing)</td>
</tr>
<tr>
<td>Span (L)</td>
<td>4800 mm (for moment resisting width)</td>
</tr>
<tr>
<td>Concrete weight</td>
<td>25.0 kN/m³</td>
</tr>
<tr>
<td>S.Wt</td>
<td>4.75 kN/m</td>
</tr>
<tr>
<td>Gross area (Ag)</td>
<td>190000 mm²</td>
</tr>
</tbody>
</table>

### Design actions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Manual</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design (M1)</td>
<td>30 D kNm/m</td>
<td></td>
</tr>
<tr>
<td>Design (M2), M1+M2</td>
<td>22.5 kNm/m</td>
<td></td>
</tr>
<tr>
<td>Ast req’d</td>
<td>472 mm²</td>
<td></td>
</tr>
<tr>
<td>Ast</td>
<td>565 mm²</td>
<td></td>
</tr>
<tr>
<td>Reinf req’d</td>
<td>N12-230</td>
<td></td>
</tr>
</tbody>
</table>

### Reinforcement

**Bottom reinf:** N12-200 cts

- Bar size = 32 mm
- Bar cts/Nm² = 200 mm
- Yield strength (fyk) = 500 MPa
- Ductility class = A (Normal) (Low) (Auto) (Auto)
- Reinf ductility class = N (Normal)
- Steel area (As) = 565 mm²/m
- Bottom cover to steel = 30 mm
- Depth to bottom steel layer (d max) = 154 mm

**Top reinf:** SL82 (main wires in main direction)

- Bar size = 16 mm
- Bar cts/Nm² = 200 mm
- Yield strength (fys) = 500 MPa
- Ductility class = A (Normal) (Low) (Auto)
- Reinf ductility class = L (Normal) (Low)
- Steel area (As) = 227 mm²/m
- Top cover to steel = 29 mm
- Depth to top steel layer = 24 mm
After inputting the basic design values, the loading for the analysis needs to be linked and input. This is done by clicking the [Analysis] button of to the side of the design page (see below). This will create a linked analysis document (Ana), which will be used to determine the moments and deflections for the slab.

Design actions

On the analysis loading document, input the span type and loadings. For this example, a superimposed dead load 0.5kPa (kN/m) and live load of 1.5kN/m is used. Note that this design is of a 1000mm wide slab strip. The span type is “S” representing a simply supported span, with the span length as 4800mm. Also ensure that the correct live load type has been set. The section properties are automatically calculated. Also note the Include S.Wt (self weight) is included and can be optionally removed.
After the load input is complete, you will see the summary of actions and graphs representing moment, shear and gross immediate deflection. The deflections shown on this document do not represent the long-term concrete deflection (creep etc). This is calculated in a later step.

The maximum moment then needs to be transferred to the design document. This is done by clicking the [Max M+*…] button on the main document (see below) which transfers the maximum moment and position this occurs at (x) from the linked loading document.

Note that for a simply supported member, we are concerned with the positive sagging moments which will occur mid-span. There are no negative hogging moments for this case. If there were, we would be also considering these by clicking the [Max M-*…] button. We will see this used in a later example of a double span slab.

Note that a moment from a different location of the slab can be selected using the “Analysis values =” input box shown below along with the “Position of result (x) =” input box shown on the previous page in the analysis/loading document.

With the moment transferred, the moment capacity, cracking stresses and minimum steel requirements need to be verified.

Section: (Slab 50x1) 190mm thick slab, f'c=32MPa
Reinf: 6/92 (main wires in main direction) top, N12-200 cts bottom, k0 = 0.08
Strength: (true M) M* = 24.6kNm < eMcr = 55.3kNm
Cracking: fcr = 225 MPa < fcr1 = 225MPa & fcr1 = 299MPA < fcr1 = 400MPa, crack width = 0.2mm
Ast/min: Ast.min = 311mm² < Ast = 565mm² (Minimum of Deemed and actual)

Interior environment with cedd b* = 600x10⁶, ccs* = 570x10⁶

Concrete strength [f'c] = 32 MPa
Depth (D) = 190 mm
Web width (Wx) = 5 mm, (t)ab
Slab type = 0 (One way, Two way & (Cl)ts (T)wo way & walls, Two-way (F)ooting

Concrete weight = 25.0 kN/m²
S.Wt = 4.75 kN/m
Gross area (Ag) = 15000 mm²

Analysis: Simple beam at midspan (Max tve M)

If the section is over capacity, the initial geometry and reinforcement can be adjusted accordingly.
After verifying the moment capacity, crack stresses and minimum steel requirements, the next step is to verify the deflections. To do this, select the [Defl] tab at the bottom of the main document. Click [Max Deflection...] to transfer the results and position of the location of maximum deflection, and [Transfer Reinf’...], to transfer the current reinforcement configuration specified into the table shown.

These reinforcement areas may need to be manually adjusted depending on whether the reinforcement is different at the left side, middle and right sides of the span; and if the compression reinforcement is outside the compression zone, therefore not acting to stiffen the slab (kcs).

As can be seen below the SL82 mesh (N7.6-200mm) was transferred to the “Top reinf’” section, and the N12-200mm to the “bottom reinf’”. As there is only a positive sagging moment in the centre of the slab (simply supported), the “Ast =” only designates reinforcement to this middle segment with the left and right ends not effective.
Note that there is a warning for $\sigma_{cs}$ shown at the top in red. This is to alert the user to ensure the correct environment has been selected in the [Creep & Shrink] tab at the bottom – refer Clause 3.1.7.2. This variable will affect the $\text{Mcr (Cracking)}$ value.

Another important factor to note is the $\text{kcs}$ value (Clause 8.5.3.2), which is used for the long-term deflection calculation. It is a function of the ratio of compression steel to tension steel. This value is limited to 0.8 (when the compression steel area is greater or equal to the tension steel area). Increasing the amount of compression steel above the tensile steel area yields no benefit to long-term deflection with the $\text{kcs}$ approach. The value of $\text{kcs}$ is 2.0 when there is no compression steel contribution. Compression steel must be in compression and a warning is provided when the compression steel is in the tension zone there-fore being ineffective for $\text{kcs}$.

$$\text{kcs} = [2 \cdot 1.2 \cdot |\text{Asc}/\text{Ast}|] \geq 0.8 = 1.519 \text{ kcs at position x}$$

For the example shown, the calculated deflection is slightly above what is desired in this case. To improve the performance, either the slab thickness must be changed, or the addition of greater compression steel which reduces the $\text{kcs}$, decreasing long-term deflections (providing the compression steel is not in the tension zone).

The area of compression steel can be changed two ways; by either manually editing the transferred reinforcement in the [Defl] tab (see below), or by changing the mesh in the [Design] tab and then clicking [Transfer Reinf’t...] in the [Defl] tab again. Ensure that if any change is made to the document or linked document that the [Max Deflection...] button is pressed again to re-verify.

For the example shown, the calculated deflection is slightly above what is desired in this case. To improve the performance, either the slab thickness must be changed, or the addition of greater compression steel which reduces the $\text{kcs}$, decreasing long-term deflections (providing the compression steel is not in the tension zone).

### Top Reinforcement:

<table>
<thead>
<tr>
<th>Top Reinforcement</th>
<th>Ast req’d</th>
<th>Design Ast</th>
<th>Ast</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Ast</td>
<td>290</td>
<td>290</td>
<td>290</td>
</tr>
<tr>
<td>Ast</td>
<td>0</td>
<td>290</td>
<td>0</td>
</tr>
<tr>
<td>N5-200</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
</tbody>
</table>

### OR

**Top Reinforcement = SL92 (main wires in main direction)**

- **Bar size**: 8.6 mm
- **Bar cts/mm²**: 200 mm
- **Yield strength (f_y)**: 500 MPa
- **Ductility class**: A (Normal)
- **Reinf’ct ductility class**: L (Normal)
- **Steel area (Asc)**: 290 mm²/m

By changing the top reinforcement to SL92, the deflection is now acceptable, as seen below.

### Section

- **[Slab SL01]** 190 mm slab, $f_c=32$ MPa

### Reinforcement

- **SL92 (main wires in main direction)** top, N12-200 cts bottom (Additional reo specified)

### Deflection

- $\delta_{cl} = 6.8$ mm, $\delta_{ll} = 1.0$ mm, $\delta_{inc} = 8.6$ mm, $\delta_{total} = 13.6$ mm (span / 353) OK
- **$\sigma_{cs}$ for interior environment with $e_{cd} = 800 \times 10^{-6}$, $e_{cs} = 570 \times 10^{-6}$**
Double Span One-way Slab

Using the same process as the single span, a double span one-way slab can be designed using a few extra steps.

Like with the single span, the trial geometry and reinforcement are input. It would be reasonable to assume that for the same span, there will be a reduction in thickness. A double span results in the same maximum \( w \cdot L^2/8 \) moment as a single span except this occurs at the middle support. The mid-span moment will reduce. Because of the reduction in thickness, the total moments as a result of dead load also reduces.

For this example, we will try using a 170mm thick slab with N12-200mm top and nominal SL82 bottom for negative bending, and N12-300mm bottom and SL82 top in positive bending. Top cover will be 20mm, and bottom cover will be 30mm as pre the previous example.

Note that the bottom reinforcement would generally be run continuously through, however, for simplicity we will maintain the reinforcement shown below.

The above values are input as with the single span. Firstly the negative bending is reviewed. Enter N12-200mm in the top and SL82 in the bottom. The document then needs to be linked to the Analysis (Ana) loading document, by clicking the [Analysis...] button.

In the linked document, input the loading configuration. This is similar to the single span, except, in order to represent a double span, the “Span type” needs to be changed to propped (P) – see below. This will result in force and deflection diagrams that represent one span of a double span slab. Note that a propped cantilever results in the same moments and deflections as a double span of equal spans.
Once the loading has been input and the span type set as "P", the maximum negative (hogging) moment can be transferred to the main document by clicking [Max M-...]. Note that the reactions for this segment (shown in the Analysis/Loading document) only represent one of the spans in isolation. This means that the middle support’s reaction is half of its true double span reaction.

Once the section has been verified for moment capacity and cracking stress and minimum steel, the maximum positive moment needs to be checked. To do this, the reinforcement needs to be changed to the positive (sagging) moment location. This is done by manually editing the current document – in this example the top reinforcement is changed to SL82, and the bottom to N12-300mm.

Alternatively, the main document can be copied and edited separately, meaning each case is assessed on different documents. This can be done by right clicking on the document tab at the top and selecting [Copy]. The suggestion is to do this once the design has been fully completed.
Once the reinforcement has been swapped, the maximum positive moment needs to be transferred by clicking the [Max M+] button on the main document’s [Design] tab. This will then allow the verification of the moment capacity, cracking stresses and minimum steel in the sagging section. Once the strength and cracking limits of both moment cases have been checked and any adjustments required made, the deflection can then be verified. Set the top and bottom bars to the maximum size (On the [Design] tab) to ensure appropriate cover and depth, N12 in this case – see below. Once done, move to the [Defl] tab.

Bottom reinfl’t = N12-300 cts

<table>
<thead>
<tr>
<th>Bar size =</th>
<th>12 mm</th>
<th>Mesh...</th>
<th>Bar size =</th>
<th>12 mm</th>
<th>Mesh...</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bar cts/No/mm² =</td>
<td>300 mm</td>
<td></td>
<td>Bar cts/No/mm² =</td>
<td>200 mm</td>
<td></td>
</tr>
<tr>
<td>Yield strength (fsy) =</td>
<td>500 MPa</td>
<td></td>
<td>Yield strength (fscy) =</td>
<td>500 MPa</td>
<td></td>
</tr>
<tr>
<td>Ductility class =</td>
<td>A (N)ormal,(L)ow,(A)uto</td>
<td></td>
<td>Ductility class =</td>
<td>A (N)ormal,(L)ow,(A)uto</td>
<td></td>
</tr>
<tr>
<td>Reinfl’t ductility class =</td>
<td>N (N)ormal,(L)ow</td>
<td></td>
<td>Reinfl’t ductility class =</td>
<td>N (N)ormal,(L)ow</td>
<td></td>
</tr>
<tr>
<td>Steel area (Asc) =</td>
<td>377 mm²/m</td>
<td></td>
<td>Steel area (Asc) =</td>
<td>565 mm²/m</td>
<td></td>
</tr>
<tr>
<td>Bottom cover to steel =</td>
<td>30 mm</td>
<td></td>
<td>Top cover to steel =</td>
<td>20 mm</td>
<td></td>
</tr>
</tbody>
</table>

Click [Max Deflection...] on the [Defl] tab to transfer the results and position of the location of maximum deflection. Check that the moments have been transferred correctly as shown below.

<table>
<thead>
<tr>
<th>Manual (M*) =</th>
<th></th>
<th>Left</th>
<th>At x</th>
<th>Right</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Manual (Ms*) =</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Analysis (M*) =</td>
<td>0.0</td>
<td>12.8</td>
<td>-22.9</td>
<td>kNm</td>
<td></td>
</tr>
<tr>
<td>Analysis (Ms*) =</td>
<td>0.0</td>
<td>9.4</td>
<td>-16.7</td>
<td>kNm</td>
<td></td>
</tr>
</tbody>
</table>

Next, transfer the reinforcement by clicking [Transfer Reinfl’t] on the [Defl] tab. However, as the [Design] tab where the reinforcement is taken from only represents one segment of the slab (which is where the design was last performed which could be a negative or positive moment region). These reinforcement areas will need to be manual corrected to represent all segments. To do this, make note of the area of reinforcement for the left, middle and right sides both top and bottom and input them as shown below.

SL82 Mesh – 227mm²

N12-200 – 565mm²

N12-300 – 377mm²

SL82 Mesh – 227mm²

Top reinfl’t:

<table>
<thead>
<tr>
<th>Ast req’d =</th>
<th>0</th>
<th>0</th>
<th>385</th>
<th>mm²/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Ast =</td>
<td>227</td>
<td>227</td>
<td>565</td>
<td>mm²/m</td>
</tr>
<tr>
<td>Ast =</td>
<td>0</td>
<td>227</td>
<td>565</td>
<td>mm²/m</td>
</tr>
<tr>
<td>-</td>
<td>N12-500</td>
<td>N12-200</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Bottom reinfl’t:

<table>
<thead>
<tr>
<th>Ast req’d =</th>
<th>0</th>
<th>229</th>
<th>0</th>
<th>mm²/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Ast =</td>
<td>377</td>
<td>377</td>
<td>227</td>
<td>mm²/m</td>
</tr>
<tr>
<td>Ast =</td>
<td>0</td>
<td>377</td>
<td>227</td>
<td>mm²/m</td>
</tr>
<tr>
<td>-</td>
<td>N12-300</td>
<td>N12-500</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Depending on the geometry of the slab, the compression reinforcement (coloured in blue) may be in tension, and therefore provide no compression capacity. In this case a warning will appear like shown below. To amend this, remove the reinforcement highlighted in red by deleting the steel area.

Leaving these erroneous areas will result in a kcs that is unconservative.

| Analysis (M*) | 0.0 | 12.6 | -22.9 | kNm |
| Analysis (Ms*) | 0.0 | 9.4 | -16.7 | kNm |

**Top reinforcement**

| Ast req'd | 0 | 0 | 384 mm²/m |
| Design Ast | 565 | 565 | mm²/m |
| Ast | 0 | 227 | mm²/m |

| N12-500 | N12-200 |

**Bottom reinforcement**

| Ast req'd | 0 | 226 | 0 mm²/m |
| Design Ast | 227 | 227 | mm²/m |
| Ast | 0 | 277 | mm²/m |

| N12-300 | N12-500 |

**Concrete**

- Un-cracked NA = 85 mm
- Un-cracked Lx = 1000 mm
- luncrk = uk0.121 W0.5/12 = 409 mm
- Tensile steel (Ast) = 377 mm²/m
- Depth to ds = 144 mm (from comp. face)
- Cond. steel (Asc) = 227 mm²/m
- dc = 56 mm (from comp. face)
- Cracked x = 0.206 mm (from top)
- Depth to cracked NA = x'ds = 0.0, 23.7, 29.6 mm (from top)

**Use Comp. steel**

- No

**Error - set comp. reinforc. = 0mm² (in tension)**

**Reinforcement Loss**

| \( b_0 \) = N12-1.5 | NOC | 0.05 |

**Tension steel ratio**

\( \rho_c = (\rho_{cm} - \rho_{cm})' \frac{W}{A} \)

**Steel ratio**

\( \rho_{cm} = \rho_{cm} / (W/A) \)

**Concrete**

- \( f_{ct, f} \) = 3.19 MPa
- Fct = 3.19 MPa
- \( f_{ct,c} \) = 12.9 kN/mm²
- CC 8.5.1.2

**Compression steel**

- \( f_{ct, c} \) = 120 kN/mm²
- CC 8.5.3.1

**Summary**

- \( kcs = 2[1.2(Asc/Ast)] = 0.8 \)
- 1.277 kcs at position x
- CC 8.5.3.2
Once a design has been found that works for moments, crack stresses and deflections, often it can be optimised to be more cost effective (or provide alternative options). In the example used for the double span, the deflection is well below the desired limit. This means that an aspect of the slab can be altered to save on costs.

For the example used, the slab’s thickness could be decreased to 150mm. Once the change has been made, the new capacities and reinforcement will need to be checked by applying the entire method detailed in the previous section – refer to the next page for a detailed flowchart.

By reducing the slab to 150mm the negative capacity can be met by using N12-175mm centres rather than N12-200mm centres

**Multi-span One-way Slab**

In a similar procedure, multi-span slabs (more than two-spans) can also be designed. This is accomplished using the Analysis Lite or Standard to determine the moments for each span. Each span is then entered into the analysis/loading document using the Span-Type = ‘O’ with a given Left end (M1*) or Right end (M2*) or both and inputting the loadings exactly as per the Analysis Standard model. The moment graphs are then matched before moving on to the design capacity step.

Use the [Max M-…] and [Max M+…] buttons or select a “position x” to design the capacity.

Deflections can also be designed with a similar approach to the double-span.

Refer to Analysing a Two Span slab tutorial for more detailed procedures.
Cantilever Spans

Cantilever spans should also be used with caution. The right support is fixed and therefore does not rotate. If the cantilever span forms part of a continuous multi-span slab (or beam) then there will be rotation at the support. The resulting cantilever deflections could either be less (as a result of the cantilever support rotating up); or more problematically more (as a result of the cantilever support rotating down). The amount this magnifies can be assessed by using Analysis Lite or Standard. Once the configuration is finalised, copy the model and fix the cantilever support to determine the effect of the back spans.

Optimising

The process of designing a double-span one-way slab can be simplified into the following flowchart.