RESERVOIR V0.0

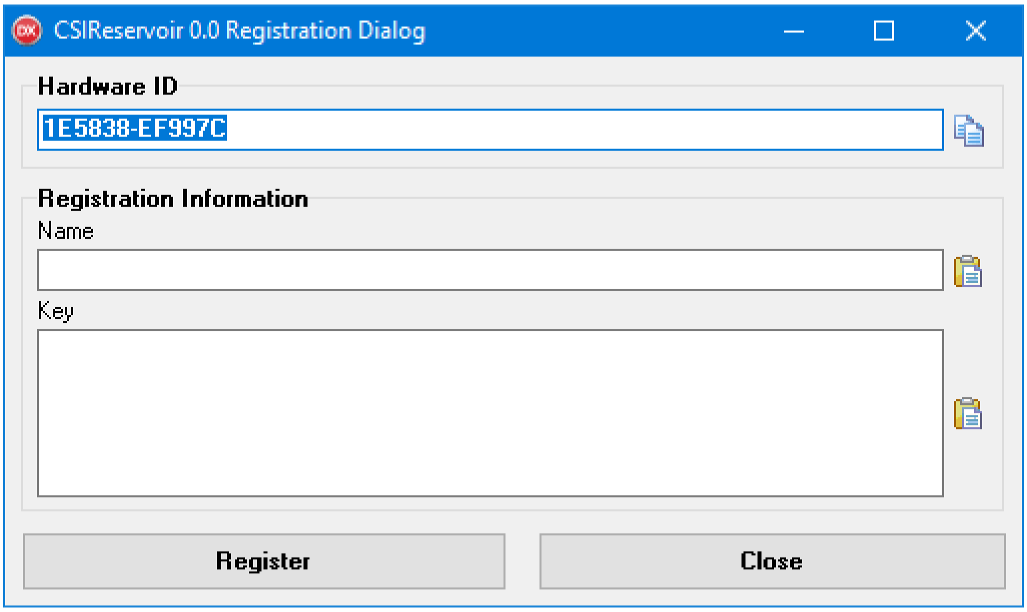
User and Reference Manual

1 Contents

2 Installation and Licensing

After unzipping the application file in your computer, you should execute **CSIreservoir32.exe** for the first time.

The following registration dialog form will pop up.



You must now send by email the Hardware ID to our licensing department and you will receive either back a normal or trial license.

This license will be a pair of strings, namely the **Registration Name** and the **Key**. Paste the received license information in the respective text boxes of the registration dialog and press **Register**.

You license should be activated now. In case it is not, use our licensing and support line to solve the problem.

3 - System Requirements

The minimum and recommended system requirements are the following:

- Operating System:

  Microsoft® Windows XP with Service Pack 2, Microsoft® Windows Vista, 32- and 64-bit versions, Microsoft Windows 7,8 or 10.

- Necessary software

  It is necessary to have in the same computer an installation of the software SAP2000 version 18,19 or 20 with the ULTIMATE level.

4 - Reservoir overview

The reservoir is a small companion tool for SAP2000 version Ultimate, aimed to help structural engineers create finite element models of ***polygonal precast post tensioned concrete panels water reservoirs***.

The goal of the software is to provide a very easy and practical way to define geometry, post tensioning stages, and other relevant events, properties and loads using a simple text file, converting it immediately into a full finite element model in ***SAP2000***.

With ***Reservoir*** the user will combine the advanced analytical capabilities of ***SAP2000*** as staged construction analysis, prestress cables, non-linear analysis, non-linear load sequences, with the practical aspect of fast model generation.

The actual version of Reservoir (V0.0 – 3rd of November of 2018) is able to generate models of SAP2000 v18, v19 and v20. It uses the SAP2000 API interface to communicate. Later versions may generate directly a $2k or s2k text file.

FIGURE 4.1

4.1 What can Reservoir consider as input?

To generate a model using ***Reservoir*** you need to use a scripting text file. This text file follows some rules. To understand these rules and the text file format, the application is bundled with 3 different examples, named EXEMPLO_1.txt, EXEMPLO_2.txt and EXEMPLO_3.txt.

Each file must have a group of sections and within those sections several variables. The explanation for each section and each variable will be given in the next chapters.

4.2 What can Reservoir produce as output?

In the actual version you can generate a SAP2000 file model, containing several entities:

a) Concrete materials for panels and slab
b) Tendon material for the cables
c) Link sections for the panel to slab connection in 2 different situations
d) Concrete shell element sections for the panels and concrete slab
e) Joint constraints to connect laterally each precast panel to the next one
f) Joint patterns for the terrain pressure, the water pressure and the later seismic acceleration
g) The shell element mesh for the whole model
h) The tendon elements to simulate the post-tensioning in the panels
i) The necessary load patterns
j) The necessary load cases, including nonlinear staged construction analysis.
k) SLS and ULS combinations

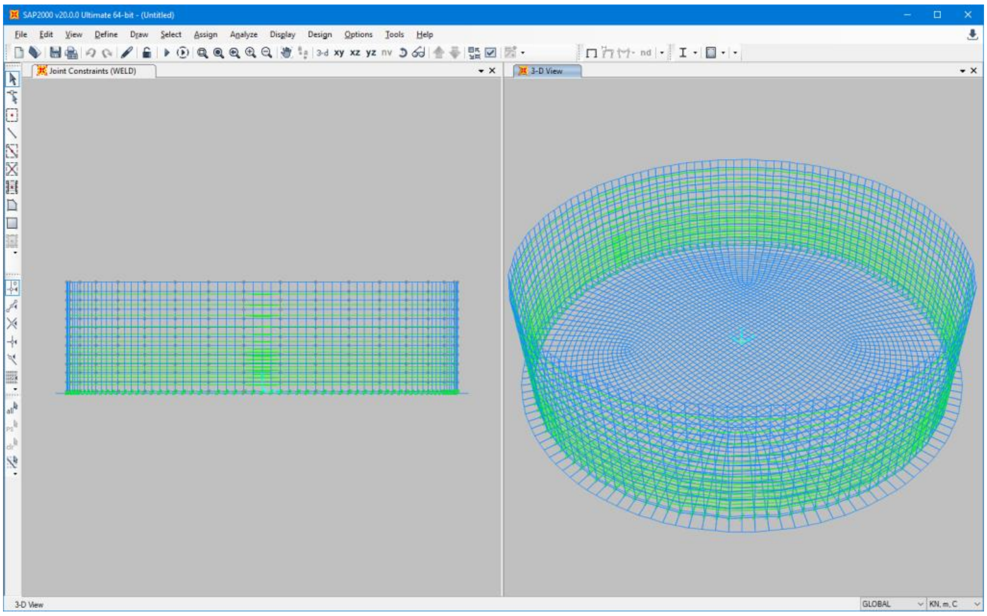


FIGURE 4.2

4.3 What is the theory used to calculate seismic effects in the reservoir?

The reservoir calculates the seismic effects based in the EN1998-8. Is assumes that the earthquake direction is along the X coordinate vector. This means that some verifications in the model results may need to be performed at different locations.

Implementation details:

Considering the reservoir as being rigid and fixed to its foundation, the horizontal seismic action can be expressed as the sum of two independent impulses: (i) “Rigid” impulse and (ii) “Convective” impulse.

For the following expressions, it’s used a cylindrical coordinate system: r, z and θ, with origin at the center of the tank bottom and the z axis vertical. The height of the tank to the original of the free surface of the fluid and its radius are denoted by H and R, respectively, p is the mass density of the fluid, while ξ = r/R and ς = z/H. The max earthquake ground acceleration is given by a_g, and the Soil factor by S. The constant β refers to the lower bound of the horizontal seismic spectrum, and has a default value of 0.20.

The total pressure is the sum of the following two impulses:

(i) The rigid impulse is given by:

$$p_i(\xi,\varsigma,\theta) = C_i(\xi,\varsigma)\rho\, H \cos\theta\, a_g\, S$$

Where:

$$C_i(\xi,\varsigma) = 2\sum_{n=0}^{\infty} \frac{(-1)^n}{I_I'(\nu_n\gamma)\nu_n^2} \cos(\nu_n\varsigma)\, I_I\left(\frac{\nu_n}{\gamma}\xi\right)$$

$$\nu_n = \frac{2n+1}{2}\pi;\;\; \gamma = \frac{H}{R}$$

I_I *and* I_I' *denote the first order modified Bessel function and its derivative*

(ii) The convective impulse is given by:

$$p_c(\xi,\sigma,\varsigma,\theta) = \rho \sum_{n=1}^{\infty} \psi_n \cosh(\lambda_n \gamma \varsigma) J_I(\lambda_n \xi)\, cos\theta\, a_g\, \beta$$

Where:

$$\psi_n = \frac{2R}{(\lambda_n^2 - 1) J_I(\lambda_n) \cosh(\lambda_n \gamma)}$$

$$J_I = Bessel\ function\ of\ the\ first\ order$$

$$\lambda_1 = 1.841,\ \lambda_2 = 5.331,\ \lambda_3 = 8.536$$

5 User guide

5.1 The application interface

The interface of the application is shown in the following screenshot:

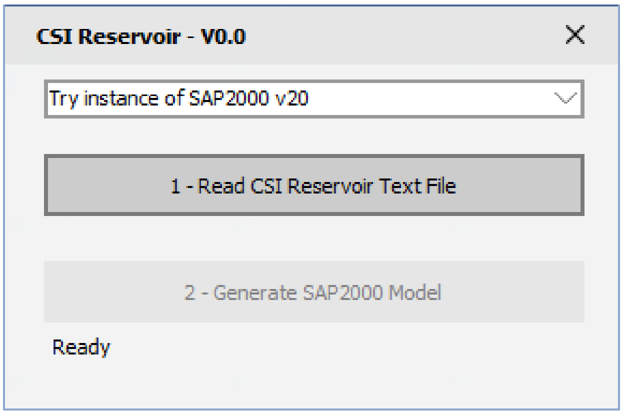


FIGURE 5.1

To use the application, you just need to perform the following steps:

1 – Select the SAP2000 version you wish to use. It will only work if that particular version is installed in your computer.

2 – Read the input text file. In case of no syntax errors you can perform the next step.

3 – Generate the SAP2000 model based in the input file information. It may take a few seconds up to a minute, depending on the size of the model. During the process, a new instance of SAP2000 will be visible, and the model will appear visible in both views of the SAP2000 interface in the end of the process in case it did work successfully.

5.2 The geometric model of the reservoir

The geometry of this template model is based in a circular slab, a polygonal panel distribution, an inner ring that may or may not have a smaller slab thickness, a panel sink distance that is used to identify the length of the panels inserted into the slab, and finally the distance between the bottom of each panel and the first tendon location.

You can see schematically the representation of this values in the figure 5.2 below.

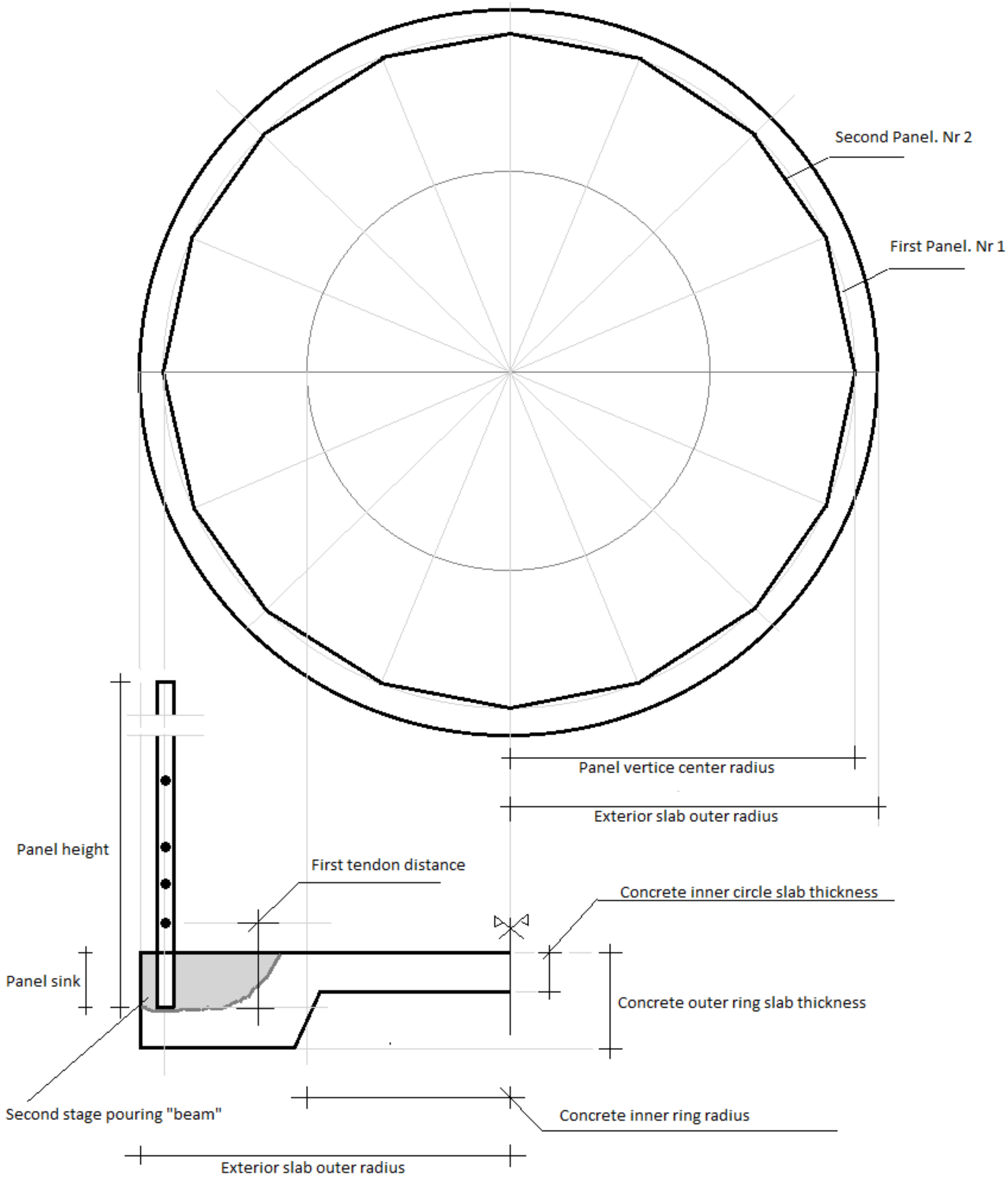


FIGURE 5.2

5.3 The input file format

The input file is divided in several blocks. Each block has a unique name. The possible blocks are:

UNITS:

MATERIALS:

SECTIONS:

TERRAIN:

GEOMETRY:

STAGES:

LOADS:

Each block contains one of more script lines. A script line is not necessarily a physical text line. A script line always starts after the end of another script line or block definition, and always ends with the character ";". This means that you can have 2 or more script lines in the same physical line.

Each script line will contain a variable assignment. This can be a single value variable or an array variable.

The script can have any number of empty lines and any number of comment lines. Comments are always preceded by "//" and must be always in any position of an empty physical text line, or the last part of any non-empty physical text line.

5.3.1 The coordinate system

In the file definition you will be requested to define certain variables that are absolute coordinates in space. For instance, the water maximum top-level Z coordinate. To have a clear understanding of what coordinates to use you must take in account this very simple and important rule:

The coordinate Z = 0 corresponds to the top face of the bottom slab.

5.3.2 The "UNITS:" block

In this block you simply define the variable "Units".

In our examples it is set like this:

```
//=====================================================================================
// Units: notice that compound units need to follow the rules of composition of units
//=====================================================================================

UNITS:

        //lb_in_F,
        //lb_ft_F,
        //kip_in_F,
        //kip_ft_F,
        //kN_mm_C,
        //kN_m_C,
        //kgf_mm_C,
        //kgf_m_C,
        //N_mm_C,
        //N_m_C,
        //Ton_mm_C,
        //Ton_m_C,
        //kN_cm_C,
        //kgf_cm_C,
        //N_cm_C,
        //Ton_cm_C
        //
        // Angles are defined in degrees

       Units:=kN_m_C;
```

Notice that there are a set of line comments allowing you to see what other options could be used in terms of units. In this example the units are set in ***[kN]*** for forces, ***[m]*** for distances and ***[C]*** for temperature (However, temperature is not used in the model, but the designation is used to keep compatibility with SAP2000).

5.3.3 The "MATERIALS:" block

In this block you define several variables.

The material for the slab concrete, named "***SlabConcrete***". In this variable you have to define the sequence of:

Ecm - Elastic Modulus [FL-2]

fck - Characteristic cylinder compressive strength

Relative humidity - [%]

cement Type - {N, R or S}

The material for the slab concrete, named "***PanelConcrete***". In this variable you have to define the sequence of:

Ecm - Elastic Modulus [FL-2]

fck - Characteristic cylinder compressive strength

Relative humidity - [%]

cement Type - {N, R or S}

The material for the tendon steel, named "***TendonSteel***". In this variable you have to define the sequence of:

E - Elastic Modulus [FL-2]

fy - The minimum yield stress [FL-2]

fu - The minimum tensile stress [FL-2]

relaxation class - {1 or 2}

The material for the soil, named "***Soil***". In this variable you have to define the sequence of:

Soil subgrade modulus - [FL-2]

Volumetric density - [FL-3]

The material for the liquid in the reservoir, named "***Liquid***". In this variable you have to define the:

Volumetric density - [FL-3]

Below you can see the example of the materials block.

```
//=====================================================================================
// Materials.
//=====================================================================================

MATERIALS:

                     // Ecm, fck, relative humidity %, cement type={N,R,S}
       SlabConcrete:= 31E06, 25E03,  80, R ;

                     // Ecm, fck, relative humidity %, cement type={N,R,S}
       PanelConcrete:= 35E06,40E03,  80, R ;

                     // E, Fy, Fu, relaxation class ={1,2}
       TendonSteel:=  195E06,1670E03,1860E03,2;

                     // Soil  subgrade Modulus,Volumetric density
       Soil:=        20E03,  19;

                     // Volumetric density
       Liquid:=      10;
```

5.3.4 The "SECTIONS:" block

In this block you define variables necessary for sections in the SAP2000 model. Namely:

Concrete outer ring thickness - [L]

Concrete Inner circle thickness - [L]

Precast panel average thickness - [L]

Tendon curvature coefficient for tendon losses - [unitless]

Wobble coefficient for the tendons – [L-1]

Anchorage Slip – [L]

Important notice: The anchorage slip that you are going to enter is prepared to correspond to a total prestress tension of 1200 MPa. During prestress operations, partial pulls will only use a fraction of this anchorage slip. However, if you aim a final different prestress tension (new final stress = FT Mpa) and for that new stress you have a defined anchorage slip you want to assume (slip = sl m), then the anchorage slip you should introduce is this block is 1200/FT x sl.

Below you can see the example of the sections block.

```
//====================================================================================
// Sections.
//====================================================================================

SECTIONS:

       ConcreteOuterRing:=   0.50;  // slab thickness (L)
       ConcreteInnerCircle:= 0.25;  // sab thickness  (L)
       PrecastPanel:=        0.164; // Panel average thickness (L)
       TendonCurvatureCoef:= 0.05;  // Tendon Curvature coef. for losses ( unitless )
       WobbleCoef:=          0.01;  // Wobble Coefficient (L-1)
       AnchorageSlip:=       4E-03; // Anchorage Set slip ( L )
```

5.3.5 The “TERRAIN:” block

In this block you only have to define a variable called “***plane***”. This is going to define the geometry of the terrain surface plane around the reservoir. To do so, you need to feed this variable with a sequence of 3 numbers:

Zmax of terrain in the surface of reservoir panels – [m]

Zmin of terrain in the surface of reservoir panels – [m]

Radial angle from Zmin to Zmax – [º Degrees]

Important notice: If no terrain is present and terrain loads should be ignored, then use “plane:=0,0,0” in this variable.

Below you can see the example of the terrain block.

```
//====================================================================================
// Terrain lateral load
//====================================================================================

TERRAIN:

       Plane:=5.5,2.0,45;     //Zmax, Zmin, radial angle of zmin to zmax
                              // use 0,0,0 for no Terrain lateral load
```

5.3.6 The “GEOMETRY:” block

This block is used to define all the geometric information about the structure. There are several variables you need to define in this block:

a) The panel height. [L]
b) The panel sink. See figure 5.2. [L]
c) The number of panels.
d) The radius of the exterior slab ring, of the panel vertice center and of the concrete inner circle [L]. See figure 5.2.
e) The anchorage panels indices. You can have 1,2 or 3 anchorage panels. You have however to defined them accordingly to the anticlockwise rule defined in figure 5.2.
f) Tendon distances [L]. This is a list of sequential vertical distances between tendons. The first distance is measured from the panel bottom to the first tendon. The next distances are always measured with respect to the previous lower tendon.
g) Tendon areas [L2]. This is a list of tendon areas, that must be in the same number of the list of tendon distances. Notice that you do not need to set a tendon area to zero if you do not wish to set it or consider it. You only need, in the STAGES block, to not define his pull force at any stage.

Below you can see the example of the geometry block.

```
//=======================================================================================
// Geometry
//=======================================================================================

GEOMETRY:

       PanelHeight:=         6.15;

       PanelSink:=           0.15;

       NrPanels:=            34;

       Radius:=              11.37,  // Exterior slab outer radius
                             10.77,  //Panel vertice center radius,
                             9.87;   //concrete inner circle radius

       AnchoragePanels:=     9,26;                  //Panels are  anticlockwise direction

       // Tendon distances between cables. First distance is from panel bottom
       // Panels are numbered in anticlockwise direction
       //                    1       2       3       4       5       6

       tendonsDistances:=    0.5,    0.2,    0.2,    0.2,    0.2,    0.2;

       TendonAreas:=         150E-6, 150E-6, 150E-6, 150E-6, 150E-6, 150E-6;
```

5.3.7 The "STAGES:" block

This block allows you to define the core aspects of the staged analysis of the reservoir. To better understand how it works, you can consult the chapter 5.4, the reservoir analyses sequences. The variables defined in this block are quite simple. However, to understand them all you need to know one very important assumption that is that ***t=0 [days] corresponds to the pouring of the bottom slab***. All other variables depend on this date.

In this block you need to define:

a) Age of panel bottom fixing by concrete pouring the fixing beam. [days]. Notice that you may have this moment coincide with the slab pouring and so you will have the panels totally embedded in the slab from the beginning. This variable is named "***tPanelFixing***".
b) The earliest moment that water or terrain loads can be applied to the structure. [days]. This variable is named "***tMinWaterFill***". To understand why there is a dual concept associated to this variable in terms of water or terrain please consult the chapter 5.4.

You also need to define an array of 3 variables. This array's length corresponds to the number of instants in time that you wish to define in terms of cable pulling. The 3 variables of this array are:

a) The age of the pull operation. [days].
b) The forces applied in each cable. [F]. Notice that these forces are also a list, and the count of this list must be the same of the number of defined cables in the GEOMETRY block. You can however have a zero value in the pull force. If a certain cable never gets any force at any stage, it will not be used in the model.
c) The type of pull in the cables. This can be either "ij", "i", or "j".
    a. "ij": cables are pulled from both ends
    b. "i": cables are pulled from the initial end
    c. "j": cables are pulled form the final end

    Notice that the start and end designation follow the order arising from using an anticlockwise angle progression in cylindrical coordinate system.

It is important to note that you may have defined 2 or 3 anchorage panels in the geometry. In this case all the 2 or 3 cables of a given level will be pulled simultaneously using the rule provided in the pull type. There is no way in this version to set the pulls of cables of the same level at different stages.

Below you can see the example of the stages block.

```
//=====================================================================================
// STAGES
//=====================================================================================

STAGES:

        // t in days

        // t=0 corresponds to the pouring of the bottom slab

        // F is force per cable in each stage

        // Pull is the indication if the tendon is pulled from start, end or from both.

        // Tendons will be defined in anticlockwise direction

        // tPanelFixing corresponds to the time when the final pouring occurs to fix the
        // panel bottom to the slab

        // tMinWaterFill corresponds to the minimum time moment that is possible that the
        // water will be filled

        t:=8;          F:=    90,    90,    90,    90,    90,    90;    Pull:=ij;

        t:=16;         F:=    90,    90,    90,    90,    90,    90;    Pull:=ij;

        tPanelFixing:=8;

        tMinWaterFill:=20;
```

5.3.8 The “LOADS:” block

This block allows you to define additional parameters necessary to the definition of load patterns in the SAP2000 model.

c) The top coordinate of the maximum water level. [L].
d) The max earthquake ground acceleration. [Lt-2].
e) The terrain lateral impulse factor. [unitless].
f) The terrain surface live load [FL-2]
g) The Soil factor, that multiplies the max ground acceleration

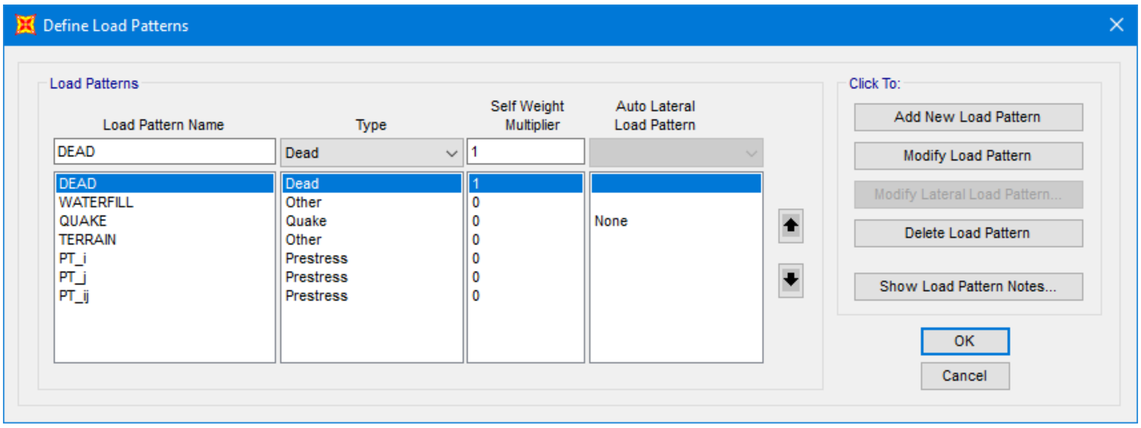
Below you can see the example of the loads block.

```
//=====================================================================================
// LOADS
//=====================================================================================

LOADS:

        WaterTopZ:=5.5;                     // water top level Z coordinate

        GroundMAxAcceleration:=2.0;         // earthquake ground acceleration (Lt-2)

        TerrainImpulseFactor:=0.55;         // Terrain lateral load stress factor

        TerrainLiveLoad:= 5.0;              // Live load over the terrain

        SoilFactor:= 2.0;                   // Soil factor
```

5.4 The reservoir analyses sequences

5.4.1 The load patterns

During the execution of the program, there are several load patterns added to the model.



a) DEAD: only the self-weight of the model is used, based on the size and property of the shell and tendon elements.

b) WATERFILL: in this load pattern shell pressure was applied based in a joint pattern named "*WaterJointPattern*". This joint pattern has the computed horizontal pressure values after taking in account the liquid volumetric weight.

c) QUAKE: in this load pattern shell pressure was applied based in a joint pattern named "*AccelerationJointPattern*". This joint pattern has the computed horizontal pressure values computed for the seismic load. See chapter 4.3.

d) TERRAIN: in this load pattern shell pressure was applied based in a joint pattern named "*TerrainJointPattern*". This joint pattern has the computed horizontal pressure values taking in account the liquid volumetric weight and the soil impulse factor.

e) PT_i : This load pattern corresponds to a cable pull in the start end of each cable. For this load pattern, a pull stress of 1200 MPa and the associated anchorage slip referenced in the input text file are set in the cable definition.

f) PT_j : This load pattern corresponds to a cable pull in the final extremity of each cable. For this load pattern, a pull stress of 1200 MPa and the associated anchorage slip referenced in the input text file are set in the cable definition.

g) PT_ij : This load pattern corresponds to a cable simultaneous pull in both extremities of each cable. For this load pattern, a pull stress of 1200 MPa and the associated anchorage slip referenced in the input text file are set in the cable definition.

5.4.2 The load cases

As a general note to the time dependent effects, the used code was the CEB-FIP90. The user can easily change in the SAP2000 interface the formulation to a different one.

5.4.2.1 Load case CONSTRUCTION

To understand the sequence of load cases, it is fundamental to observe how was the Load case associated with the construction phase defined. The load case “CONSTRUCTION” is the base of all analyses, and it represents the staging of the construction sequence.

The first stage of this load corresponds to adding all the concrete structure with sliding panels over the bottom slab. The slab “age at add” variable is set to the smallest of the ages between the secondary beam pouring moment used to constraint the bottom of the panels, and the first cable pull moment. The panels always enter in the structure with 28 days.

In the next stages, the cable pulls and the panel fixing pouring will be the necessary moments to identify independently in this load case.

Notice that when the pouring of the panel fixing occurs, then, the links connecting the panels to the slab change from sliding to fixed.

5.4.2.2 Load case LONGTERM_FILLED

This non-linear staged load case starts from the end of the CONSTRUCTION load case and it lets time pass until the moment of water fill. It then applies the water pressure and finally simulates a period of 20.000 days occurring.

Notice that in this load case, the forces resultant from the CONSTRUCTION load case will be inherited and added to the ones resulting from its own calculation.

This load case will give you an idea of the severity of the water impulse when the effects of prestress will decrease in time because of losses and material time dependent properties.

5.4.2.3 Load case LONGTERM_EMPTY

This non-linear staged load case starts from the end of the CONSTRUCTION load case and it lets time pass until the moment of terrain pressure loading. It then applies the terrain pressure and finally simulates a period of 20.000 days occurring.

Notice that in this load case, the forces resultant from the CONSTRUCTION load case will be inherited and added to the ones resulting from its own calculation.

This load case will give you an idea of the severity of the prestress and terrain impulse combined. This load case will also maximize the creep and losses effects in the prestress.

5.4.2.4 Load case PT-FINAL-FILLED

This is just a static nonlinear case define to obtain the post tensioning isolated contribution at infinite time when the reservoir lived with time losses associated with the water filling loading conditions. In this case we start from the case LONGTERM_FILLED, and we remove the water and the self-weight (by adding them with negative sign).

5.4.2.5 Load case PT-FINAL-EMPTY

This is just a static nonlinear case define to obtain the post tensioning isolated contribution at infinite time when the reservoir lived with time losses associated with the terrain loading conditions. In this case we start from the case LONGTERM_EMPTY, and we remove the terrain and the self-weight (by adding them with negative sign).

5.4.2.6 Load case PT-TRANSFER

In order to obtain the maximum post tensioning effect immediately after the end of the construction stage, you just inherit from the CONSTRUCTION load case, and subtract the DEAD load pattern in this nonlinear static analysis.

5.4.2.7 *Load cases DEAD, WATERFILL, QUAKE*

These simple linear static cases are in fact run by using the stiffness and structure of the end of the CONSTRUCTION load case. In this way, unused tendons will not be creating weight and stiffness in the model during analysis.

5.4.3 The combinations

Reservoir will also generate a set of load combinations for ULS and SLS, including to envelopes for both. These combinations as most of the other settings in the constructed SAP2000 model can be easily changed in the interface. Below is the table that represents the combinations defined for the model.

	DEAD	TERRAIN	WATTERFILL	QUAKE	PT-TRANSFER	PT-FINAL-FILLED	PT-FINAL-EMPTY
ULS1.T	1		1.35	1.5	1.2		
ULS1.F	1		1.35	1.5		0.85	
ULS1.E	1		1.35	1.5			0.85
ULS2.T	1	1.5			1.2		
ULS2.F	1	1.5				0.85	
ULS2.E	1	1.5					0.85
SLS1.T	1		1		0.85		
SLS1.F	1		1			0.85	
SLS1.E	1		1				0.85
SLS2.T	1	1			0.85		
SLS2.F	1	1				0.85	
SLS2.E	1	1					0.85