Tutorial

Heat of Hydration Analysis by Construction Stages

CONTENTS

Overview

The rate and amount of heat generation are important in concrete structures having considerable mass. A rise in temperature accompanies thermal expansion, and non-uniform cooling of mass concrete creates undesirable stresses. Thermal cracking in a concrete structure tends to be wide and propagates through the structure. This naturally has adverse effects on strength, durability and permeability. Moreover, mass concrete structures are cast in many stages with construction joints. Individually constructed segments exhibit different heat source properties and time dependent properties. Therefore, construction stages must be incorporated in a heat of hydration analysis model to truly reflect a real construction process.

Stresses due to heat of hydration are classified as **Internal Constraining Stress** and **External Constraining Stress**. The **Internal Constraining Stress** results in from the restraining effect of volumetric changes due to different temperature distributions within the concrete structure. For instance, at the initial state of hydration, temperature differences between the surface and inner parts result in surface tension. Whereas at a latter stage, contracting deformations in the inner parts are greater than those at the surface, thereby resulting in tension stresses in the inner parts. The magnitude of the Internal Constraining Stress is proportional to the temperature difference between the surface and inner parts. **External Constraining Stress** is caused by restraining the volumetric change of fresh concrete in contact with subsoil or the substrate of previously cast concrete. The change in concrete heat results in the change of volume, and the restraining effect is dependent on the contact area and stiffness of the external constraining objects.

Heat of hydration analysis can be accomplished through **Heat Transfer Analysis** and **Thermal Stress Analysis**. Heat Transfer Analysis entails the process of calculating the change of nodal temperatures with time due to heat source, convection, conduction, etc., which take place in the process of generating heat of hydration of cement. Thermal stress analysis provides stress calculations for mass concrete at each stage based on the change of nodal temperature distribution with time resulting from the heat transfer analysis. The stress calculations also account for time and temperature dependent material property changes, time dependent shrinkage, time and stress dependent creep, etc.

This tutorial demonstrates the process of construction stage analysis and analyzes the results for a foundation structure constructed in two stages or pours. The tutorial also outlines the procedure of generating a construction stage model for heat of hydration analysis and reviewing the analysis results:

Enter general material properties	Modulus of elasticity, Specific heat, Coefficient of heat conductivity
Enter time dependent material properties	Consider Creep & Shrinkage and change in modulus of elasticity
Create a structural model	Create elements, define boundary conditions & input loads
Heat of Hydration Analysis Control	Define integration factor & initial temperature. Input whether to consider Creep & Shrinkage and the calculation method. Select whether to use Equivalent Age and to consider the self weight load.
Ambient Temperature Functions ***Convection Coefficient Functions*** ***Element Convection Boundary***	After entering ambient temperature and convection coefficient functions, use them to define convection boundary conditions
Prescribed Temperature	Assign constant temperature conditions to the parts, which do not undergo any temperature changes with time
Heat Source Functions ***Assign Heat Source***	Enter heat source functions and assign them to the corresponding elements
Pipe Cooling	Enter relevant data if pipe cooling is used.
Construction Stage	Define elements, boundary conditions and load conditions corresponding to each construction stage. Set initial temperature of elements being activated.
Perform analysis	Perform heat transfer analysis and thermal stress analysis
Check analysis results	Analyze temperature distribution and variation of thermal stresses with time

* Pipe cooling is not included in this tutorial for clarity in demonstrating the interaction of the two concrete parts while analyzing the results of heat of hydration analysis.

Structural data for analysis model

This example represents a simple foundation structure often encountered in practice. It consists of subsoil mass and two parts of mass concrete cast in two stages as shown in Figure 1. The 2nd pour takes place after 170 hours of casting the 1st pour. Heat of hydration analysis is performed for the period of 930 hours after casting the 2nd concrete mass.

If the subsoil mass, that is interfaced with the concrete, is modeled as soil springs to represent the boundary condition, the transfer of the concrete heat cannot be properly represented. Therefore, we will create a model which includes the foundation having properties of specific heat and thermal conductivity, to closely represent the true behavior as shown in Figure 1.

Subsoil mass	: 24 x 19.2 x 3 m
Mat foundation (1st pour)	: 14.4 x 9.6 x 2.4 m (170 hours)
Mat foundation (2nd pour)	: 14.4 x 9.6 x 2.4 m (930 hours)
Cement type	: Low-heat of hydration cement

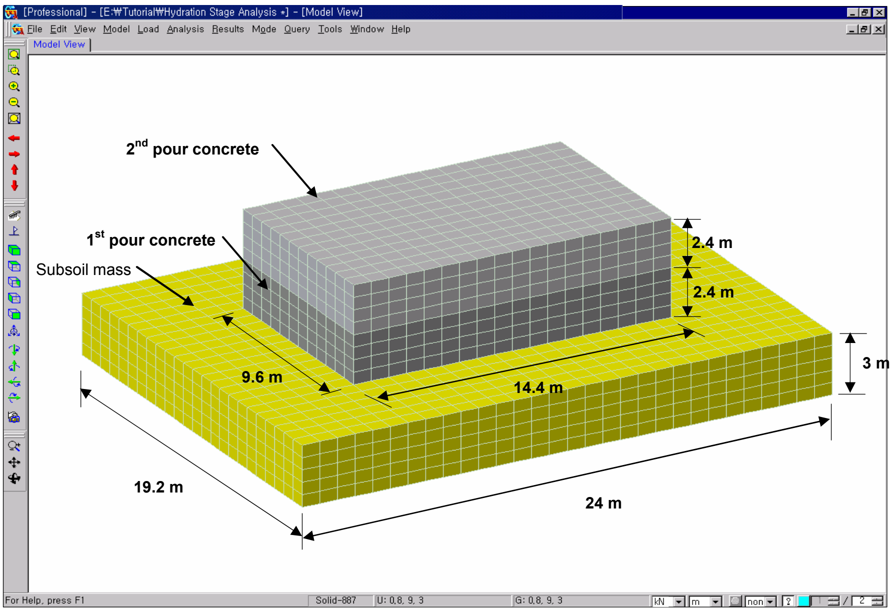


Figure 1 Heat of hydration model for construction stages

In this tutorial, due to symmetry of the structure, we will model and analyze only one quarter of the entire structure as shown in Figure 2. The use of symmetry not only reduces the analysis time, it also provides convenience in checking the internal temperature and stress distribution.

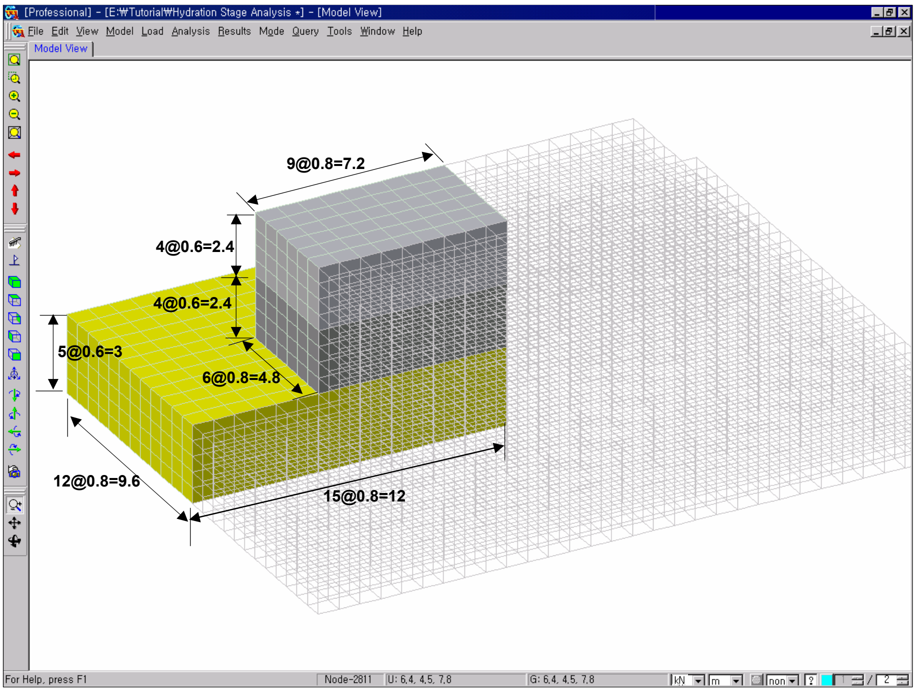


Figure 2 Heat of hydration model for construction stages (1/4 symmetry model)

Material and thermal properties

The material and thermal properties are summarized in Table 1 below.

Table 1. Material and thermal properties

Property \ Part		Lower foundation	Upper Foundation	Subsoil
Specific heat (kcal/kg ℃)		0.25	0.25	0.2
Density (kgf/m^3)		2400	2400	1800
Rate of heat conduction (kcal/m hr ℃)		2.3	2.3	1.7
Convection coefficient (kcal/m^2 hr℃)	Surface exposed to atmosphere	12	12	12
	Steel form	12	12	-
Ambient temperature (℃)		20	20	-
Casting temperature (℃)		20	19	-
91-day compressive strength (kgf/cm^2)		270	270	-
Compressive strength gain coefficients		a=13.9 b=0.86	a=13.9 b=0.86	-
91-day modulus of elasticity (kgf/cm^2)		2.7734×10^5	2.7734×10^5	1.0×10^4
Thermal expansion coefficient		1.0×10^{-5}	1.0×10^{-5}	1.0×10^{-5}
Poisson's ratio		0.18	0.18	0.2
Unit cement content (kg/m^3)		320	320	-
Heat source function coefficients		K=33.97 a=0.605	K=33.97 a=0.605	-

This example uses low heat of hydration cement. The maximum adiabatic temperature rise (K) and reactive velocity coefficient (a) are based on experimental values pertaining to the unit cement content.

Analysis modeling

Setting work environment

Open a new file (***New Project***) and ***Save*** it as **'Heat of Hydration.mcb'**.

File / ***New Project***
File / ***Save*** **(Heat of Hydration)**

Select a unit system, which is often used for thermal property data, namely m and kgf, as shown in Figure 3.

Tools / Unit System
Length>**m** ; Force> **kgf** ↵

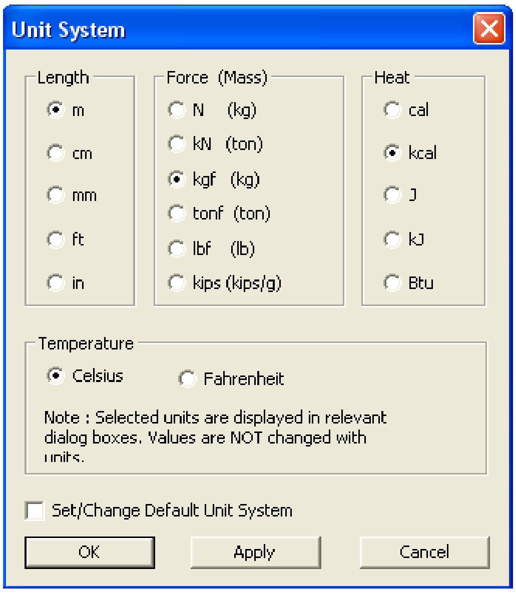


Figure 3 Assigning a unit system

Defining material properties

Define the properties of mat foundation and subsoil.

Model / Properties / **_Material_**

General>Material Number > **1** ; Name>**(Mat Foundation)** ; Type>**Concrete**

Concrete>Standard> **ASTM(RC)** ; DB>**C4000**

Thermal Coefficient>**Celsius (on)**

Thermal Transfer>Specific Heat>**(0.25)** ; Heat Conduction>**(2.3)** ↵

General>Material Number>**2** ; Name>**(Subsoil)** ; Type>**User Defined**

Modulus of Elasticity>**(1.0e+8)** ; Poisson's Ratio>**(0.2)**

Thermal Coefficient>**(1.0e-5)** ; Weight Density>**(1800)**

Thermal Transfer>Specific Heat>**(0.2)** ; Heat Conduction>**(1.7)** ↵

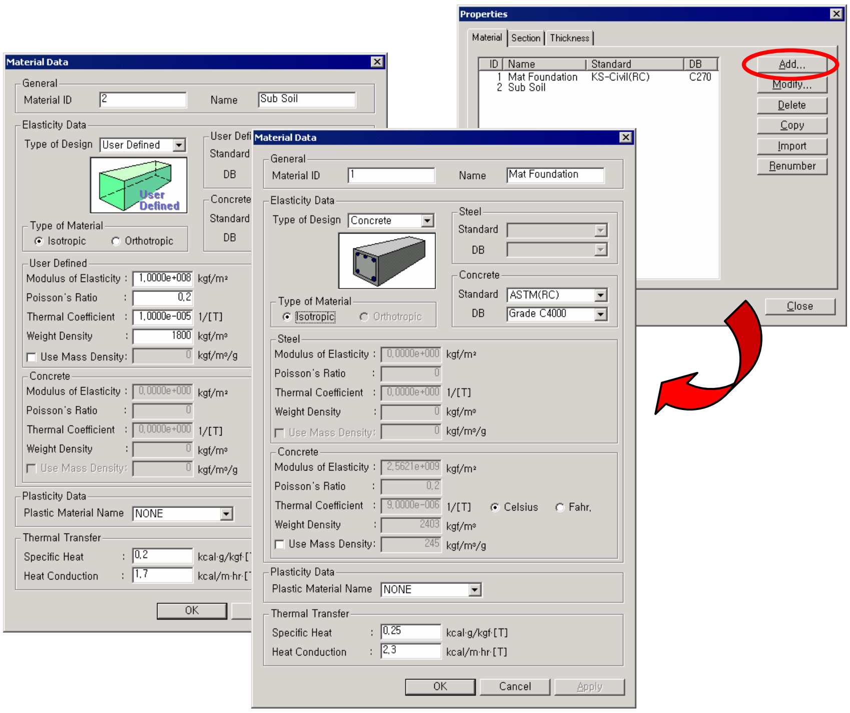


Figure 4 Defining material properties

Defining time dependent material properties

Define time dependent material properties to account for creep, shrinkage and change of modulus of elasticity.

Model / Properties / ***Time Dependent Material (Creep/Shrinkage)***

Name>**(Creep/shrinkage)** ; Code> **ACI**

Compressive strength of concrete at the age of 28 days>(**2700000**)

Relative Humidity of ambient environment (40~99)>(**70**)

Volume-surface ratio>**(0.12)**

Age of concrete at the beginning of shrinkage>**(3)**

Init Curing Method>**moist cure**

Material factored ultimate value

Type>**ACI Code** ; Slump>**(0.12)** ; Fine aggregate percentage>**(40)**

Air content>**(4)** ; Cement content>**(320)** ↵

Model / Properties / ***Time Dependent Material (Comp. Strength)***

Name>**(Elasticity)** ; Type>**Code** ; Code>**ACI**

Refer to "Using MIDAS/Civil > Model > Properties > Time Dependent Material (Elasticity)" in the On-line Manual.

Concrete Compressive Strength at 28 Days (f28)>**(2700000)**

Concrete Compressive Strength Factor (a, b)>**(13.9, 0.86)** Redraw Graph ↵

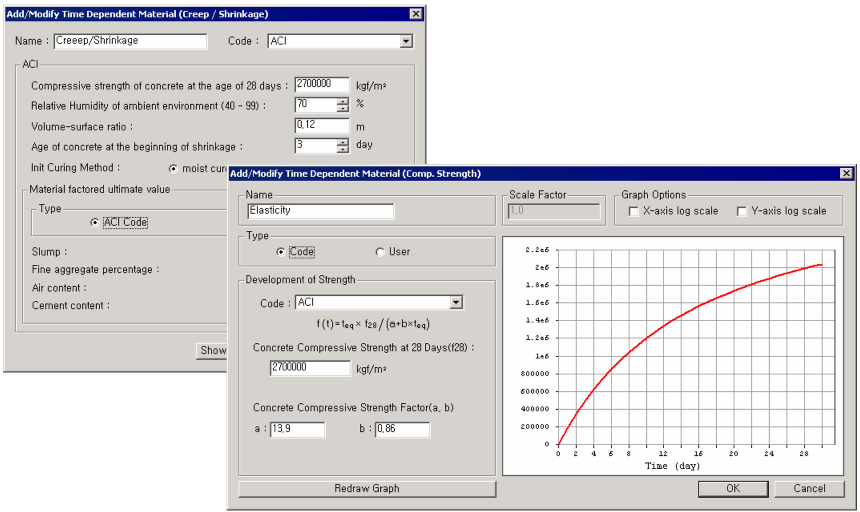


Figure 5 Defining time dependent material properties

Linking general and time dependent material properties

It is now necessary to link the previously defined general and time dependent material properties as per Figure 6.

Model / Properties / ***Time Dependent Material Link***
Time Dependent Material Type>Creep/Shrinkage>**Creep/Shrinkage**
Time Dependent Material Type>Elasticity>**Elasticity**
Select Material for Assign>Materials>**1: Mat Foundation**
Operation> Add / Modify

Even if Effective Modulus is used to consider Creep, select the Creep / Shrinkage functions and link them to general materials to assign elements for which the creep is to be calculated.

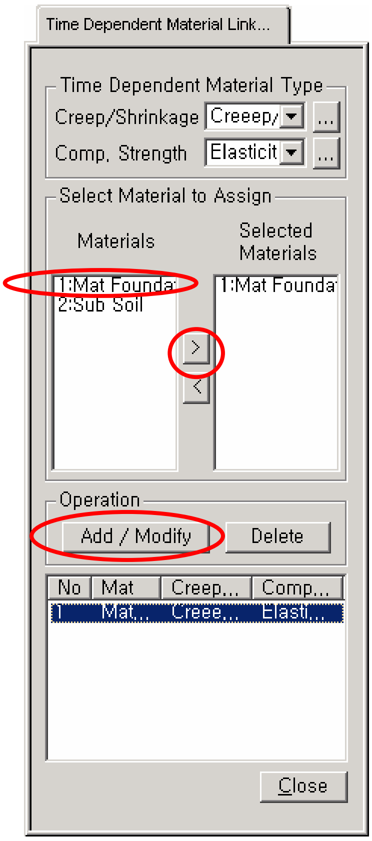


Figure 6 Linking general and time dependent material properties

Structural modeling

First, generate a plate element representing the base of the subsoil mass by creating a node at a lower corner and extending it to the remaining corner nodes. This plate element is then extruded into a solid using ***Extrude Elements.***

Point Grid (off) ; ***Point Grid Snap*** (off) ; ***Line Grid Snap*** (off)

Node Number (*Toggle on*)

Top View

Auto Fitting

Model>Nodes> ***Create Nodes***

Coordinates **(0,0,0)** ; **(12,0,0)** ; **(12,9.6,0)** ; **(0,9.6,0)**

Model>Elements> ***Create Elements***

Elements Type>**Plate**

Type>**Thick (on)**

Material>**1 : Mat Foundation**

Nodal Connectivity>**(1, 2, 3, 4**)

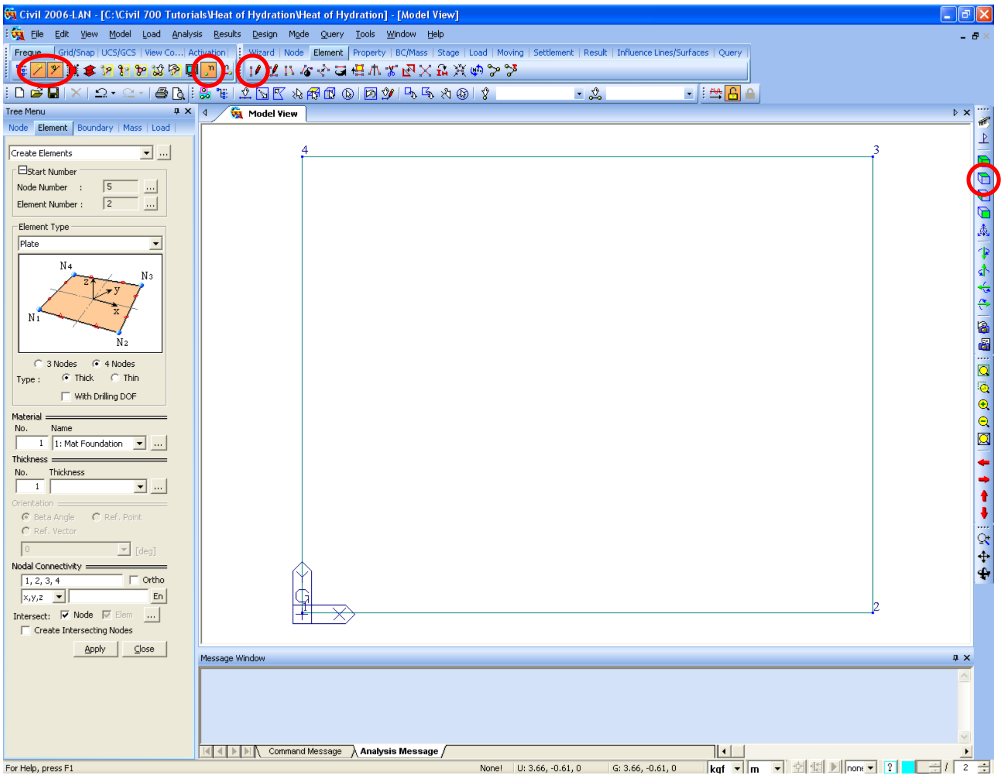


Figure 7 Creating a plate element representing the base of subsoil mass

Using ***Extrude Elements***, create a solid element.

Iso View

Model>Elements> ***Extrude Elements***

Select All

Extrude Type>**Planar Elem. → Solid Elem.**

Source>**Remove** (on)

Element Type>**Solid** ; Material>**1: Mat Foundation**

General Type>**Translate** ; Number of Times = **1**

Translation>**Equal Distance (on)** ; dx,dy,dz>**(0, 0, 7.8)** ↵

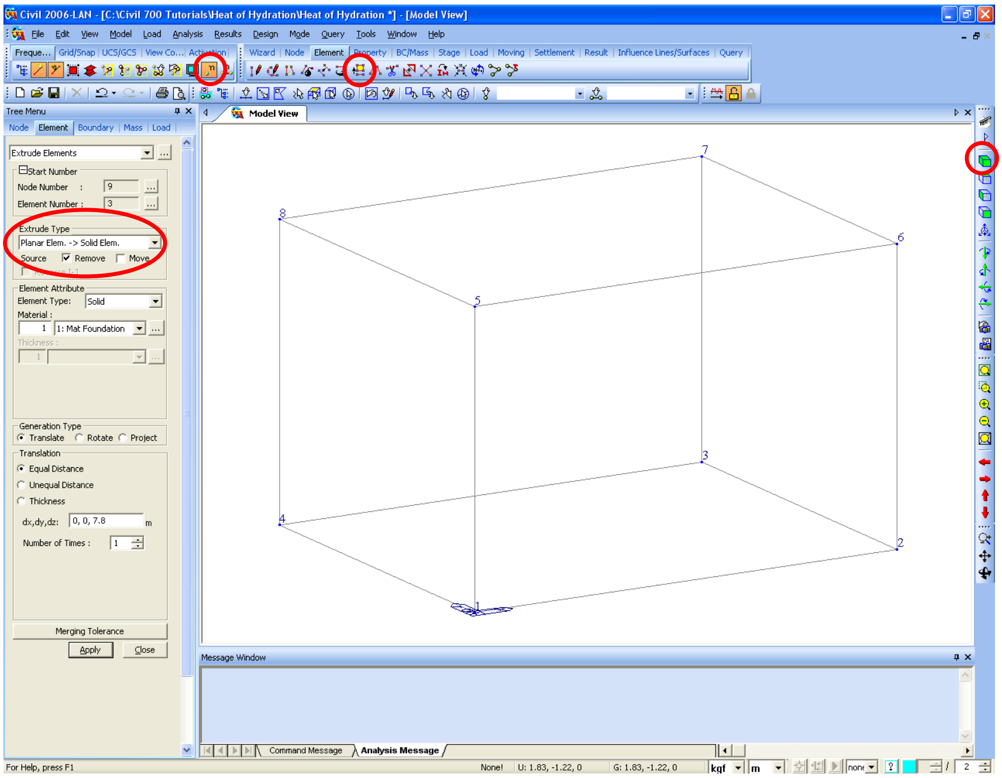


Figure 8 Creating a solid element

Division of element

Next we divide the element using ***Divide Elements.*** The size of mesh depends on the total configuration. We should also pay attention to the parts, where we anticipate significant changes in stresses, for fine meshing. The subsoil part does not need fine meshing, and yet it needs to be meshed such a way that no significant change in stresses takes place within an element. For the sake of simplicity, we will divide the element uniformly as shown in Figure 9.

Model / Elements / ***Divide Elements***

Select All

Divide Elements>Element Type>**Solid** ; **Equal Distance**

Number of Divisions x: **(15)** ; y: **(12)** ; z: **(13)** ↵

Hidden *(Toggle on)*

Node Number (*Toggle off*)

Display>Node tab>**Node** (off)

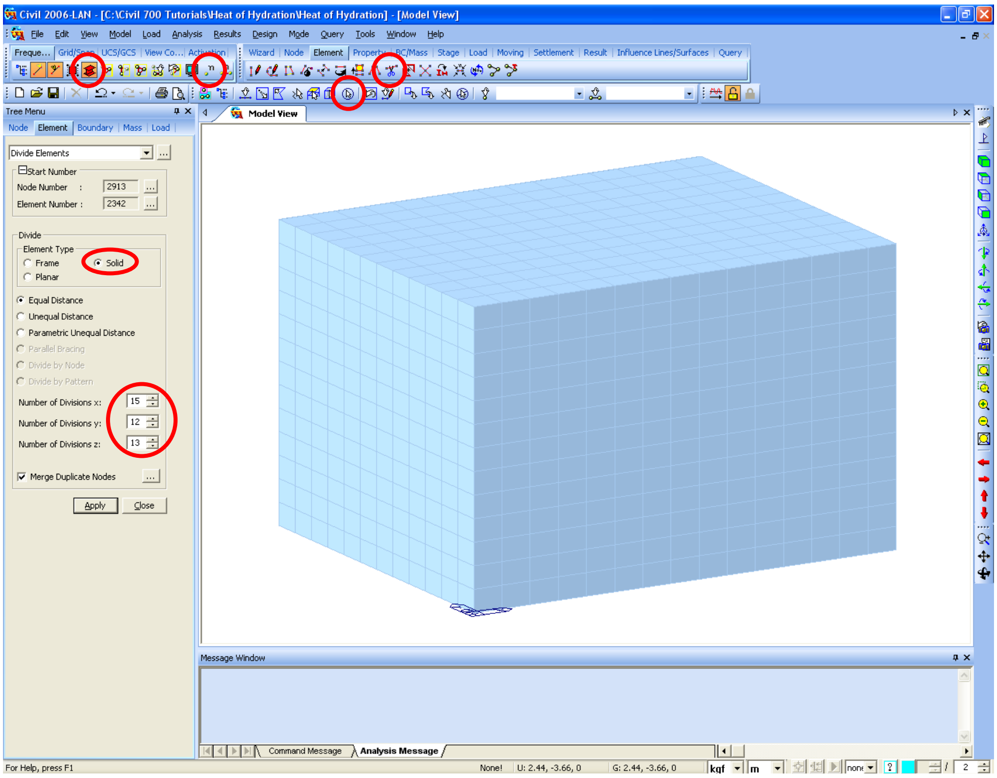


Figure 9 Division of solid element

Now that we created a mesh consisting of brick elements using ***Extrude Elements*** and ***Divide Elements***, we will now delete unnecessary elements from the overall model.

Front View

Shrink

Model>Elements> ***Delete Elements***

Select Window (① in **Figure 10**)

Type>**Selection** ; **with Free Nodes** (on) ↵

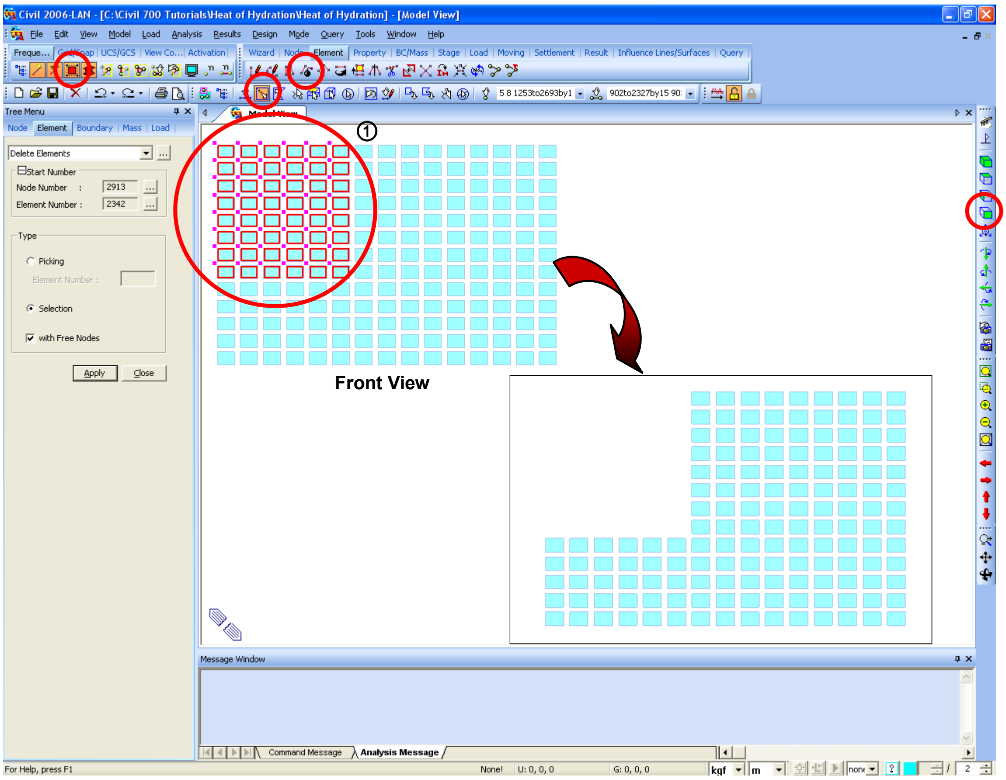


Figure 10 Deleting elements

Now change the view point to ***Left View***, and delete the elements which do not belong to the model.

Left View

Model>Elements> ***Delete Elements***

Select Window (① in **Figure 11**)

Type>**Selection** ; **with Free Nodes** (on) ↵

Iso View

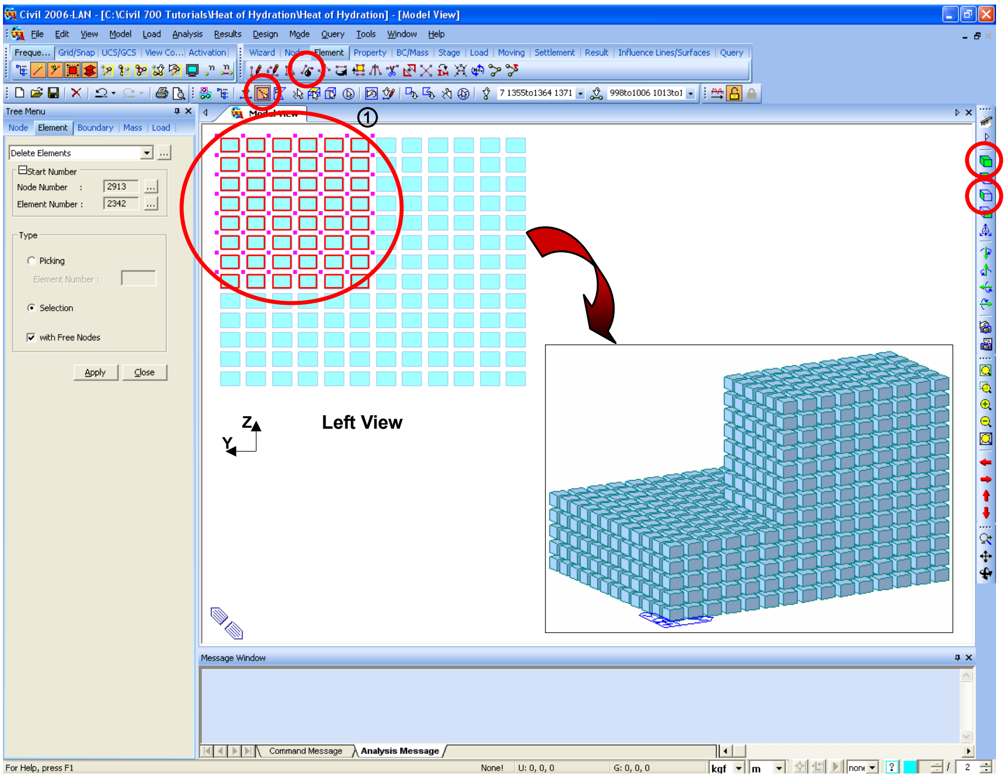


Figure 11 Deleting additional elements

When we created the 3-D solid element using ***Extrude Elements***, we assigned it as a concrete material. We will now revise the material to that corresponding to the soil material.

Change Element Parameters can be also used to change the properties of elements.

Tree Menu>Works tab

Front View

Select Window (① in **Figure 12**)

Properties>Material>**2: Subsoil (Drag & Drop)**

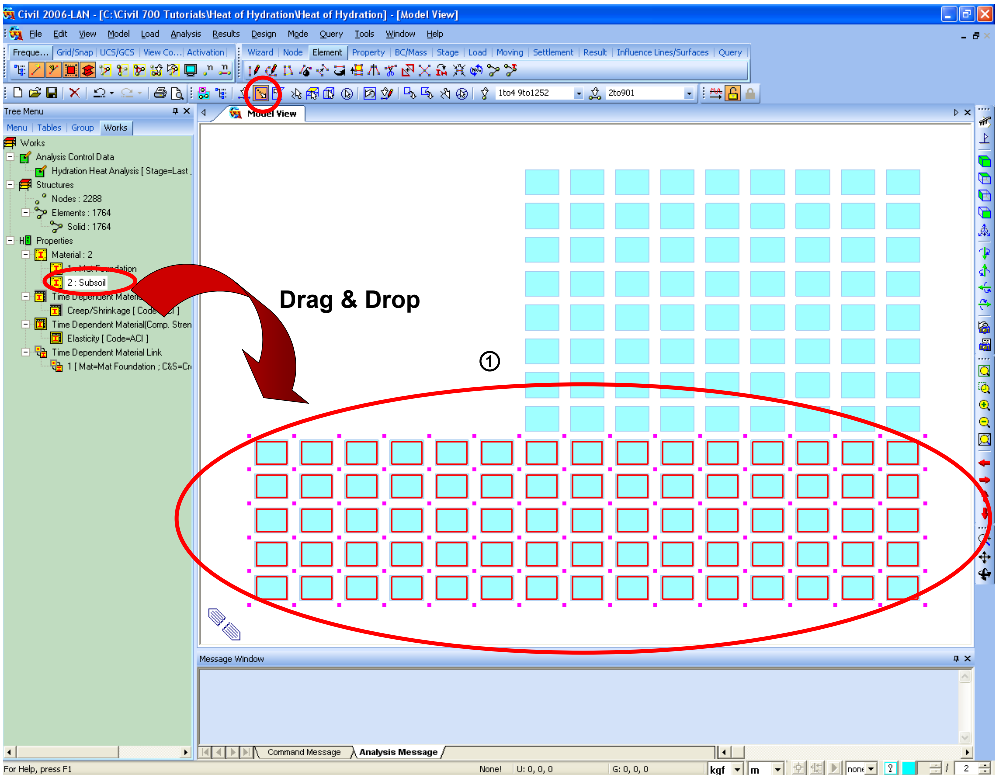


Figure 12 Assigning subsoil material properties

Defining Structure Groups

In order to perform construction stage analysis, we need to define the element and boundary condition groups that activated or deactivated at each construction stage. These groups are then used to define the construction stages. First, we create Structure Groups.

C Group>Structure Group >**New…** *(by right-click on Structure Group)*

Define Structure Group>Name>**Subsoil** ↵

Define Structure Group>Name>**Mat Foundation (Lower part)** ↵

Define Structure Group>Name>**Mat Foundation (Upper part)** ↵

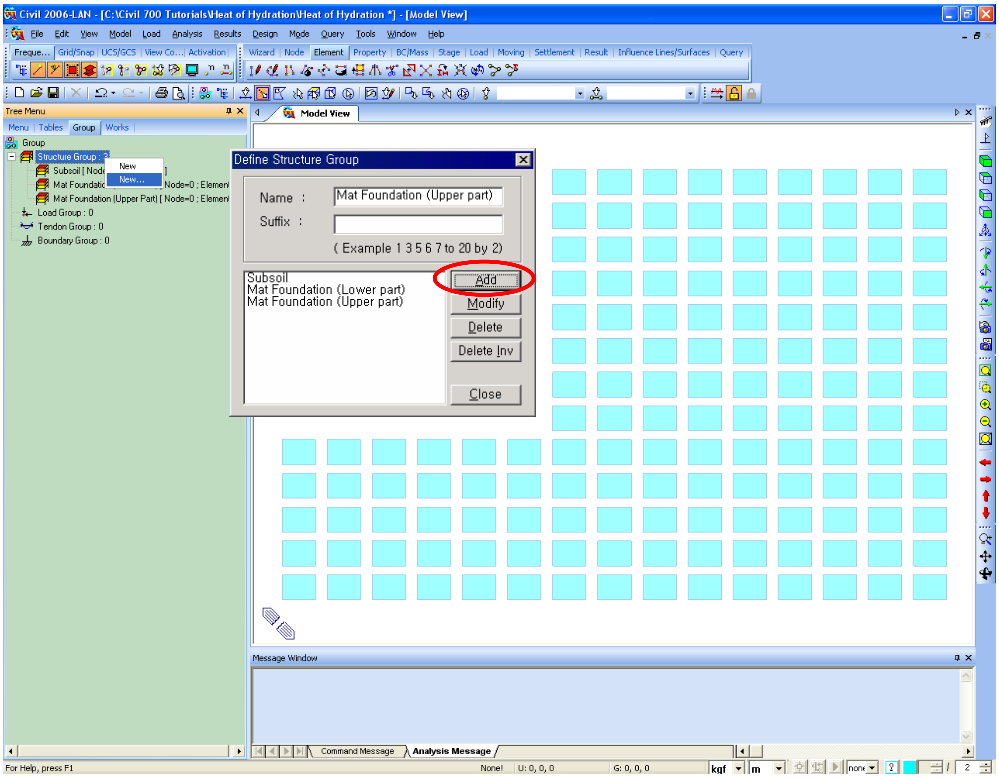


Figure 13 Creating Structure Groups

Assigning elements to Structure Groups

We now assign relevant elements to the Structure Groups created and, thus, define the Structure Groups. First, we group the elements pertaining to the subsoil into the Subsoil Structure Group.

Tree Menu>Group tab
Select Window (① in **Figure 14**)
Structure Group>**Subsoil (Drag & Drop)**

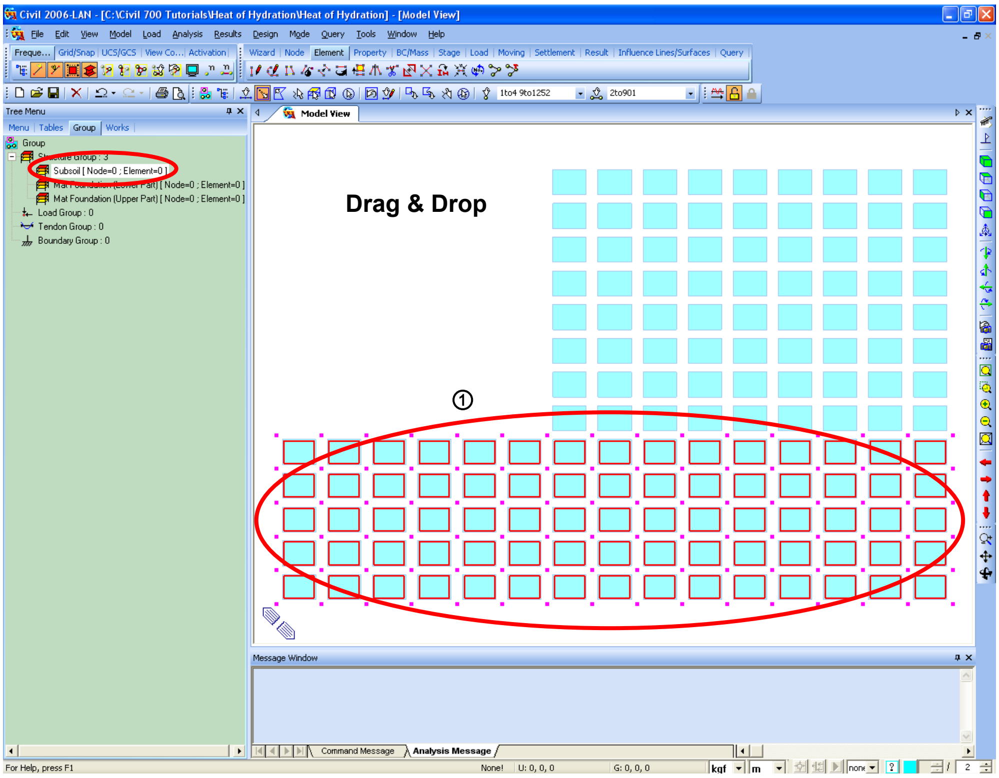


Figure 14 Defining the Structure Group "Subsoil"

Assign Structure Groups for the Mat Foundation, 1st poured lower part and the 2nd poured upper part.

Tree Menu>Group tab

Select Window (① in **Figure 15**)

Structure Group>**Mat Foundation (Lower Part) (Drag & Drop)**

Select Window (② in **Figure 15**)

Structure Group>**Mat Foundation (Upper Part) (Drag & Drop)**

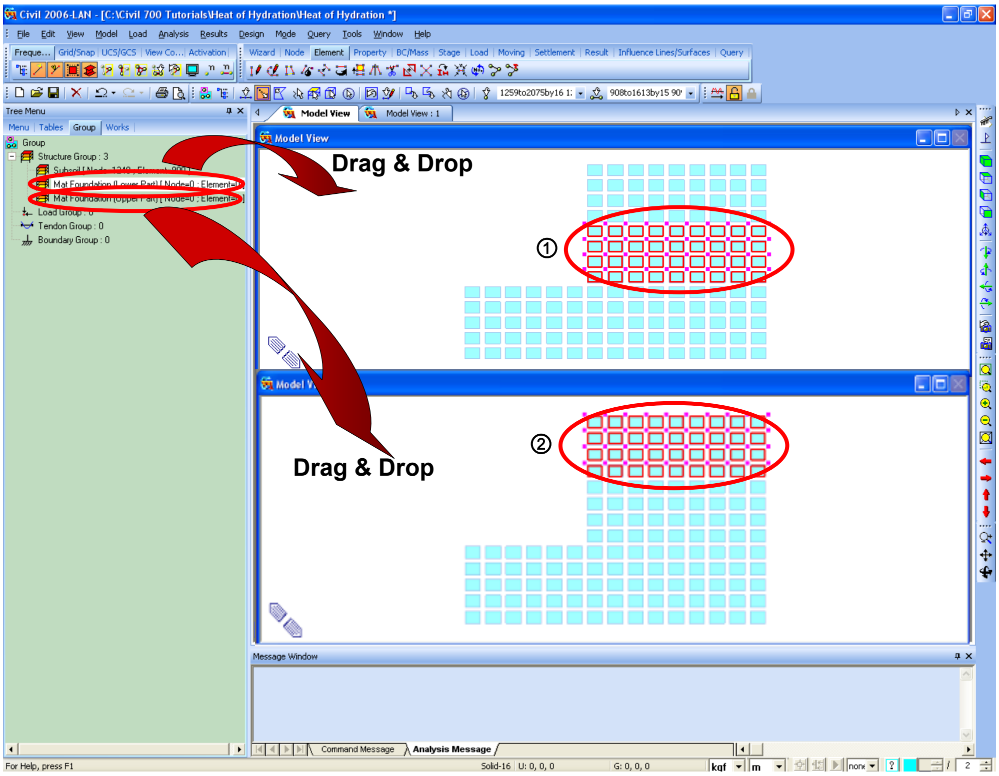


Figure 15 Defining Structure Groups for Mat Foundation (Lower & Upper Parts)

Defining Boundary Groups

We now create boundary groups as Figure 16.

Boundary Surface group represents the construction joint surface between the 1st and 2nd pours.

Group>Boundary Group >**New…**

Define Boundary Group>Name>**CS1** ↵

Define Boundary Group>Name> **CS1-Boundary Surface** ↵

Define Boundary Group>Name> **CS2** ↵

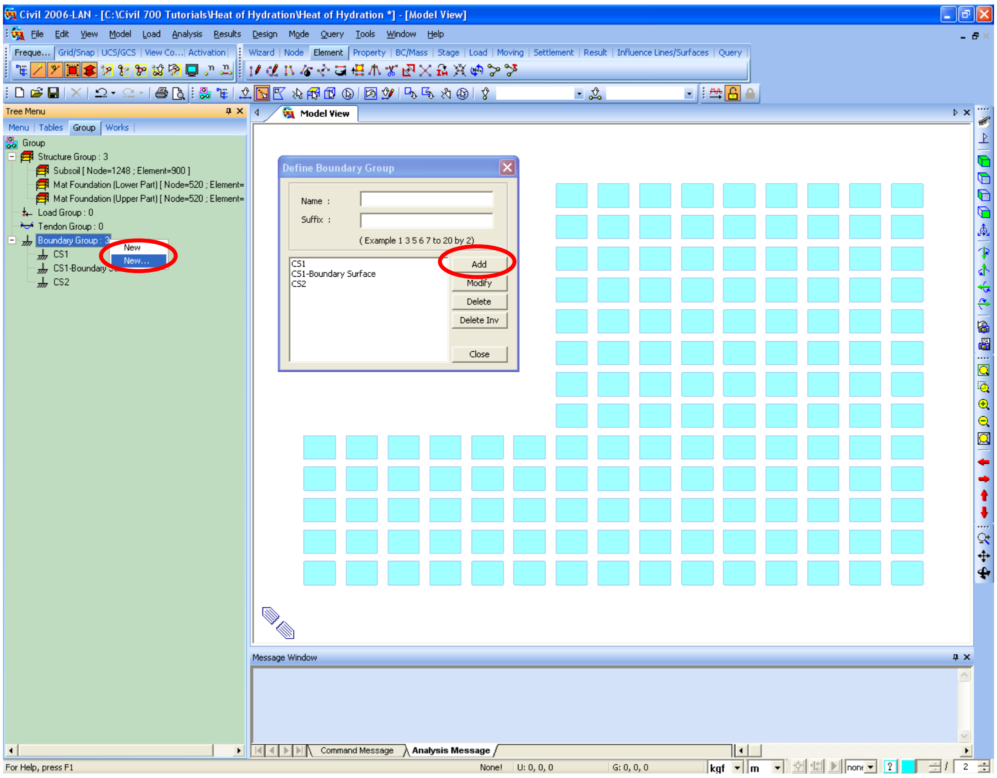


Figure 16 Creating Boundary Groups

Next, we enter the Subsoil boundary conditions for each group.

We will create a multi-window showing ***Front View*** (in Model View) and ***Left View*** (in Model View : 1) for the ease of modeling.

Window / ***New Window***
Left View ; ***Hidden*** ; ***Shrink***
Point Grid (off) ; ***Point Grid Snap*** (off) ; ***Line Grid Snap*** (off)
Model View
Window / ***Tile Horizontally***
Zoom Fit (Model View & Model View : 1)
Model / Boundary / Supports
  Select Window (① in **Figure 17**)
  Select Window (② in **Figure 17**)
  Boundary Group Name>**CS1**
  Options>**Add**
  Support Type>**D-All** (on) ↵

Solid elements do not retain rotational degrees of freedom. Therefore, we need to restrain only translational DOFs.

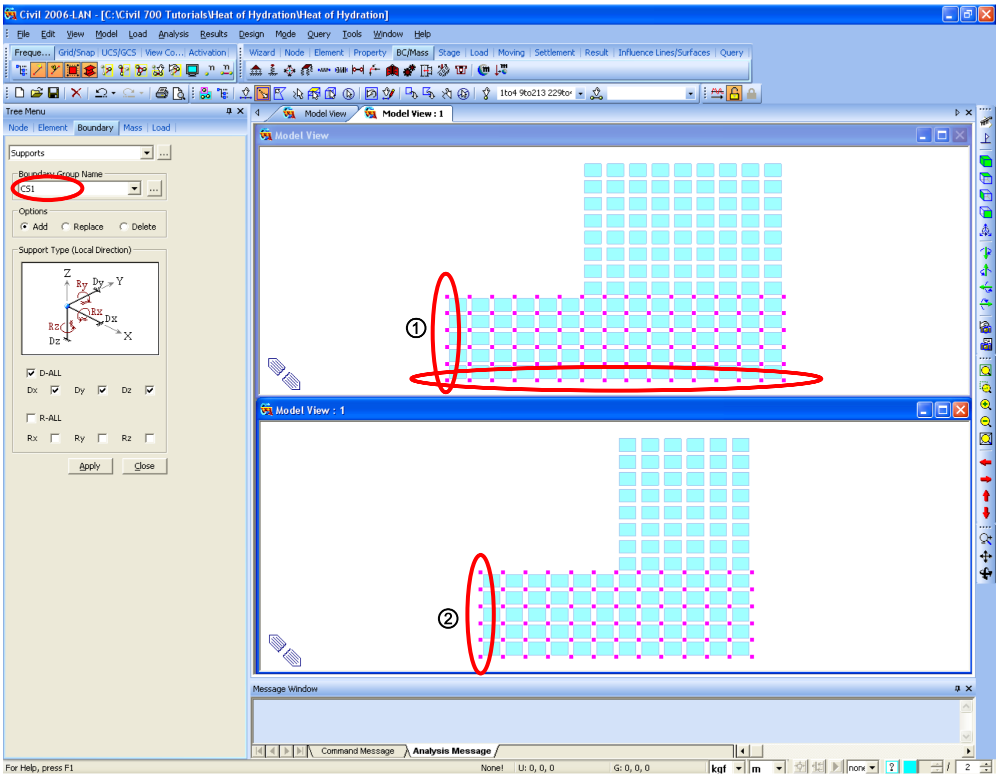


Figure 17 Defining Subsoil boundaries

Since it is a 1/4 symmetrical model, we need to specify the symmetric boundary condition. First, we will enter the symmetry condition pertaining to the 1st pour.

Model / Boundary / Supports

Select Window (① in **Figure 18**)

Boundary Group Name>**CS1** ; Options>**Add**

Support Type>**Dx** (on) ↵

Select Window (② in **Figure 18**)

Boundary Group Name> **CS1** ; Options>**Add**

Support Type>**Dy** (on) ↵

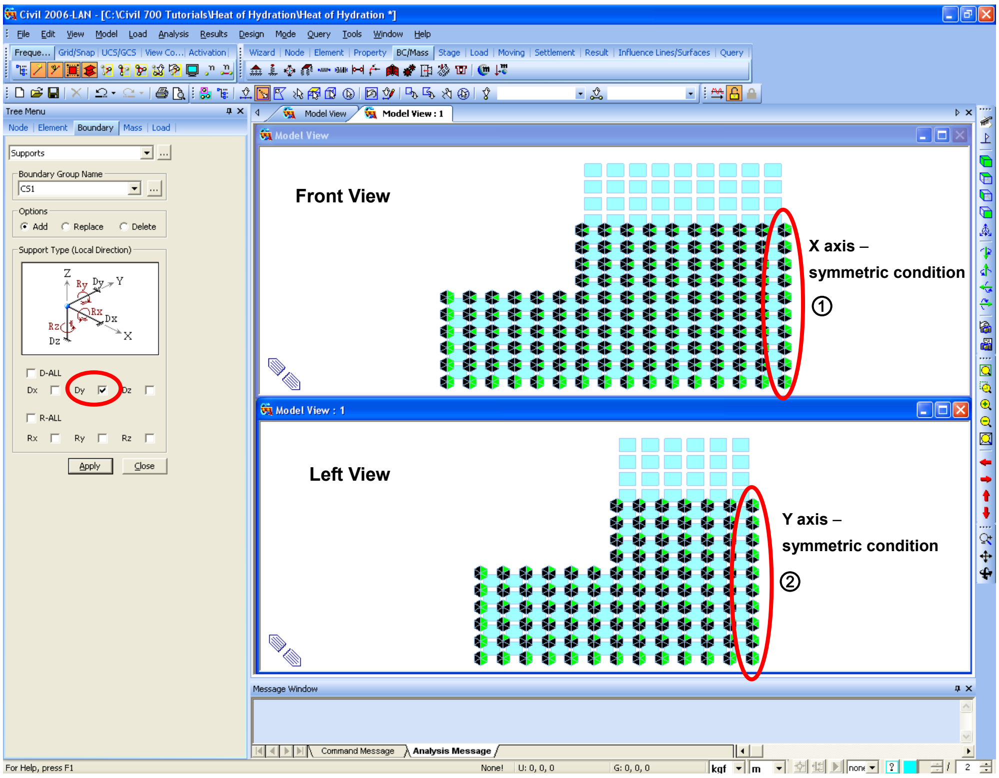


Figure 18 Entering symmetric boundary conditions

We continue on to specify the symmetric condition for the 2nd pour.

Model / Boundary / Supports

Select Window **(① in Figure 19)**

Boundary Group Name>**CS2** ; Options>**Add**

Support Type>**Dx** (on) ↵

Select Window **(② in Figure 19)**

Boundary Group Name> **CS2** ; Options>**Add**

Support Type>**Dy** (on) ↵

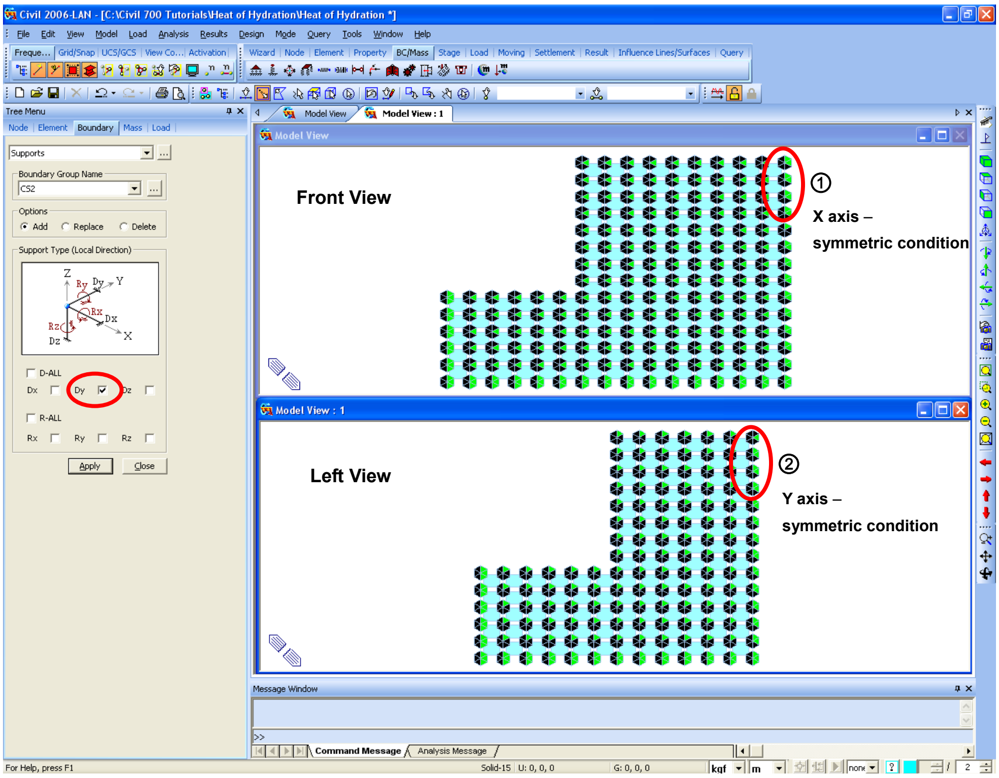


Figure 19 Entering symmetric boundary conditions

Defining Load Group

Define Load Groups and Load Cases to include static load in the construction stages of heat of hydration analysis. Static load cases entered in heat of hydration analysis must be input as Construction Stage Load Type.

Model View> ***Maximize***

Group>Boundary Group >**New…**

Define Boundary Group>Name>**Self** ↵

Load>Static Load Cases

Name>**Self** ↵

Type>**Construction Stage Load (CS)** ↵

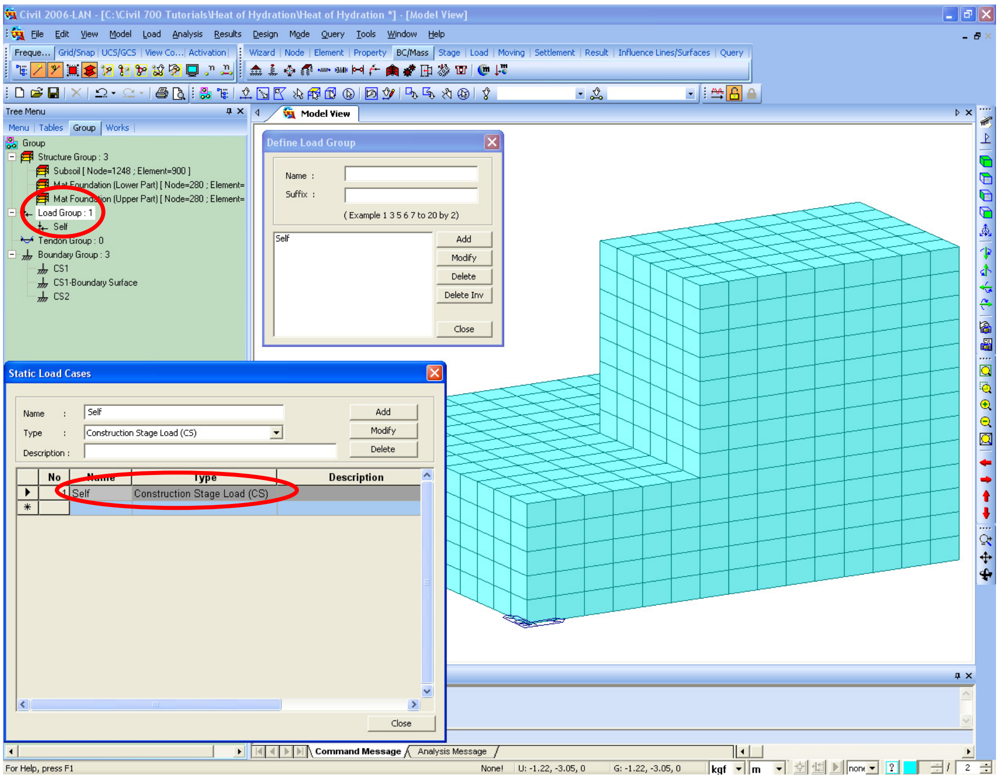


Figure 20 Definition of Load Group & Load Case

Heat of hydration analysis can consider static load cases for construction stage analysis. First, self weight is assigned.

Load>Self Weight
Load Case Name>**Self**
Load Group Name>**Self** ↵
Z : **(-1)** ↵

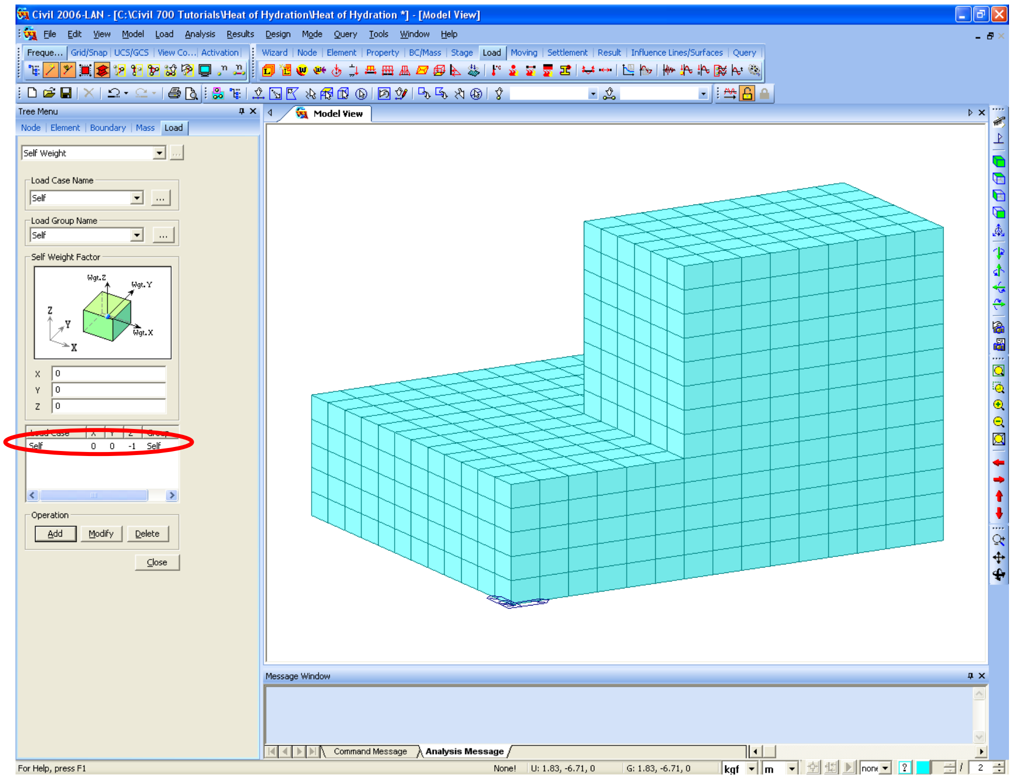


Figure 21 Inputting self weight

Inputting heat of hydration analysis data

For assigning the conditions for analysis, refer to "Heat of Hydration Analysis" in Analysis for Civil Structures and the Online manual – Using MIDAS/Civil > Analysis > Hydration Heat Analysis Control.

Heat of Hydration Analysis Control

Now that the analysis model is completed, we will enter the required data noted below (time integration factor, initial temperature & stress output location) for heat transfer analysis.

Analysis / ***Heat of Hydration Analysis Control***

Final Stage>**Last Stage**

Integration Factor>**(0.5)**

Initial Temperature>**(20)**

Element Stress Evaluation>**Gauss**

Creep & Shrinkage (on) ; Type>**Creep & Shrinkage**

Creep Calculation Method>**General**

Number of Iterations=**5** ; Tolerance=**0.001**

Use Equivalent Age by Time & Temperature (on)

The Initial Temperature can be superseded by the values entered in the Compose Construction Stage for Hydration dialog box.

If creep is to be considered by reducing the modulus of elasticity without using general creep functions, select Effective Modulus.

If a general creep function is to be used, define the function and select General.

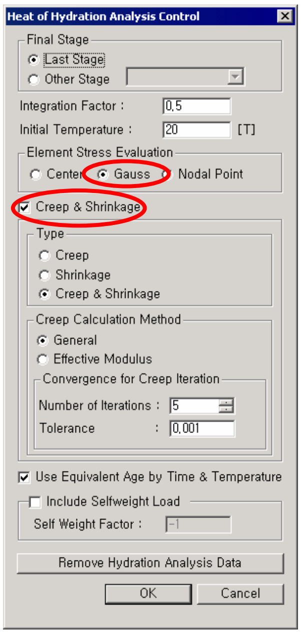


Figure 22 Data entry for Heat of Hydration Analysis Control

Inputting ambient temperature

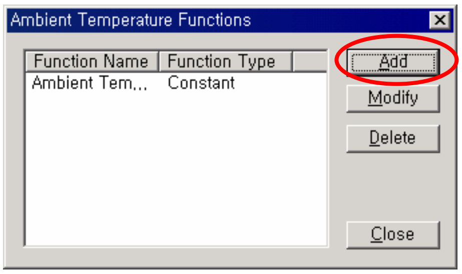
Ambient temperature is now entered as a function of time. This example assumes a constant temperature of 20℃.

Load / Heat of Hydration Analysis Data / ***Ambient Temperature Functions***

Function Name>**(Ambient Temperature)**

Function Type>**Constant**

Constant>Temperature>**(20)** ; Redraw Graph ↵



Select User type and enter the Time and Temperature variations, if they are not constant.

If ambient temperature varies at different locations due to exposure to the atmosphere, being partly immersed in water, etc., a number of Ambient Temperature Functions can be defined and applied.

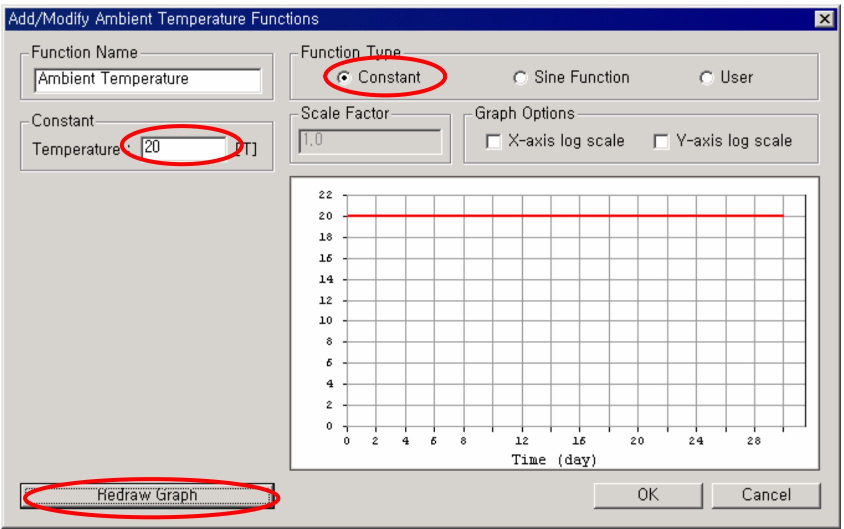


Figure 23 Entering ambient temperature function

Inputting convection coefficient

Next we enter the convection coefficient as a function applicable at the concrete surface.

User type can be used if the heat exchange condition between the concrete surface and the atmosphere varies with time due to the change in curing conditions.

Load / Heat of Hydration Analysis Data / ***Convection Coefficient Functions***

Function Name>**(Convection Coeff)**

Function Type>**Constant**

Constant>Convection Coefficient>**(12)** ; Redraw Graph ↵

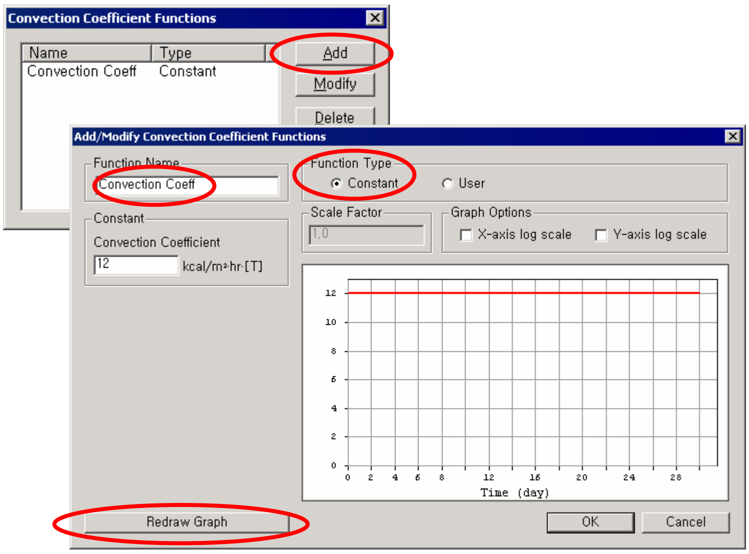


Figure 24 Entering convection coefficient function

Boundary Surface group represents the construction joint surface between the 1st and 2nd pours.

We now assign the previously defined ambient temperature and convection coefficient function to the concrete surface, which is exposed to the atmosphere. Depending on the construction stages, the surface exposed to the atmosphere changes as well. Accordingly, we assign the corresponding ambient temperature and convection boundary conditions to the previously defined CS1, CS1-Boundary Surface and CS2. First, we assign the ambient temperature and convection coefficient to the concrete surface exposed to the atmosphere at the time of 1st pour. Since the concrete surface between the 1st and 2nd pours will not be exposed to the atmosphere at the time of the 2nd pour, it is defined as another group.

Window / ***New Window***

Window / ***Tile Horizontally***

Load / Heat of Hydration Analysis Data / ***Element Convection Boundary***

Select Window (① in **Figure 25**)

Select Window (② in **Figure 25**)

Boundary Group Name>**CS1**

Option>**Add/Replace**

Convection Boundary>Convection Coefficient Function>**Convection Coeff**

Ambient Temperature Function>**Ambient Temperature**

Selection>**By Selected Nodes** ↵

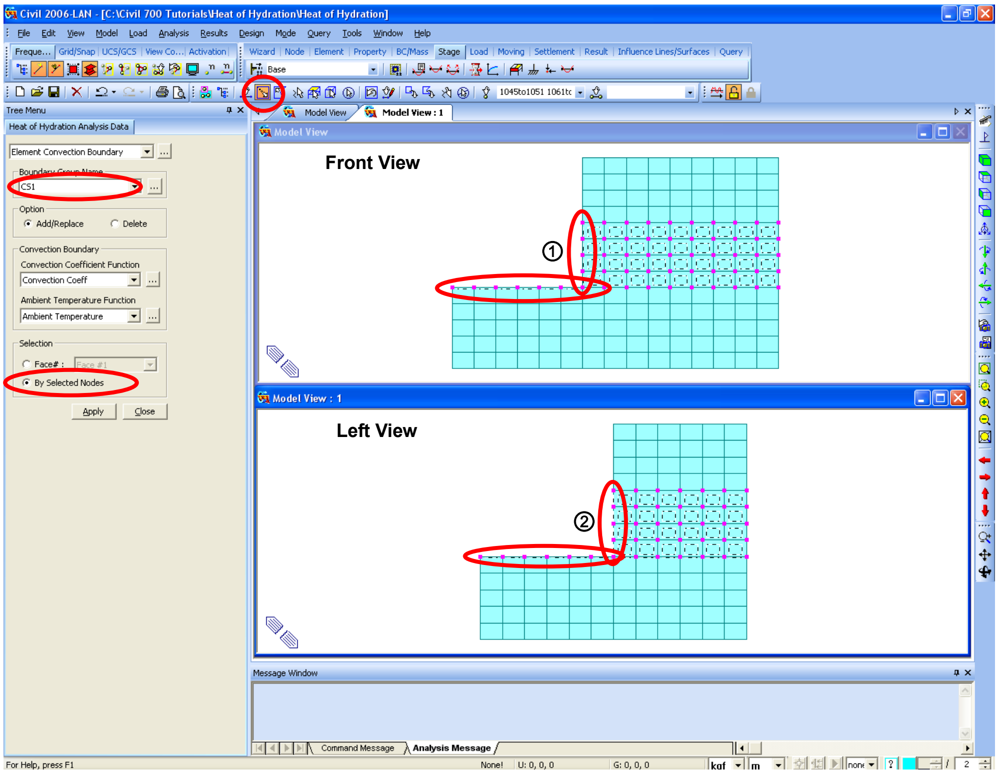


Figure 25 Defining convection boundary at the 1st pour stage

We now define the convection boundary condition at the surface joining the 1st and 2nd pours.

Load / Heat of Hydration Analysis Data / ***Element Convection Boundary***

Select Window (① in **Figure 26**)

Boundary Group Name>**CS1-Boundary Surface**

Option>**Add/Replace**

Convection Boundary>Convection Coefficient Function>**Convection Coeff**

Ambient Temperature Function>**Ambient Temperature**

Selection>**By Selected Nodes** ↵

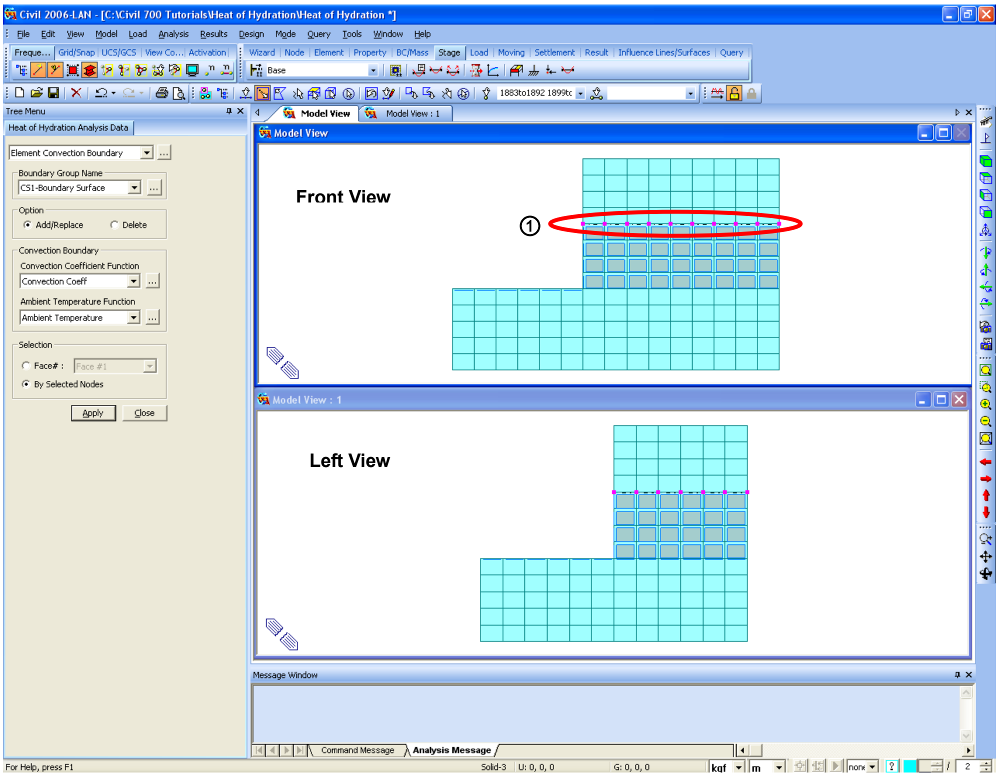


Figure 26 Defining convection boundary condition at the boundary surface

We now move on to define the convection boundary surface of the 2nd pour.

Load / Heat of Hydration Analysis Data / ***Element Convection Boundary***

Select Window (① in **Figure 27**)

Boundary Group Name>**CS2**

Option>**Add/Replace**

Convection Boundary>Convection Coefficient Function>**Convection Coeff**

Ambient Temperature Function>**Ambient Temperature**

Selection>**By Selected Nodes** ↵

Select Window (② in **Figure 27**) ↵

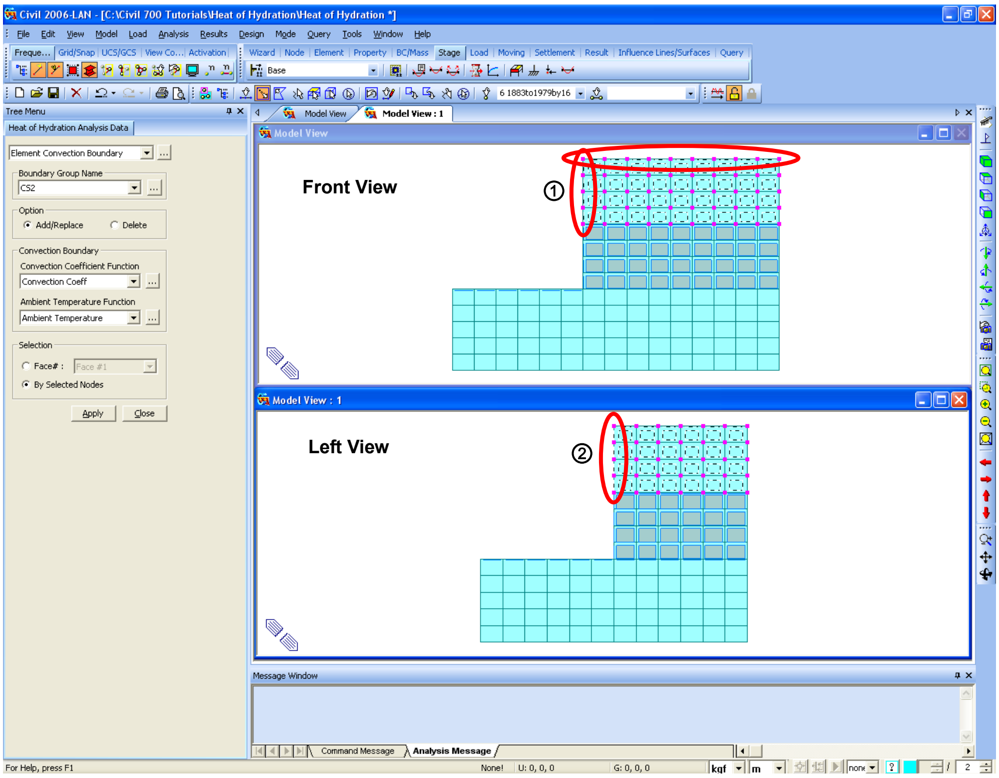


Figure 27 Defining the convection boundary condition at the 2nd pour stage

Defining constant temperature condition

We enter a constant temperature condition for those parts where temperature remains unchanged. Assign a constant temperature to those surfaces, which have not been assigned the symmetric boundary condition or the convection boundary condition (for example, boundary surface in contact with the soil).

Load / Heat of Hydration Analysis Data / ***Prescribed Temperature***

Select Window (① in **Figure 28**)

Select Window (② in **Figure 28**)

Boundary Group Name>**CS1**

Option>**Add**

Temperature> Temperature **(20)** ↵

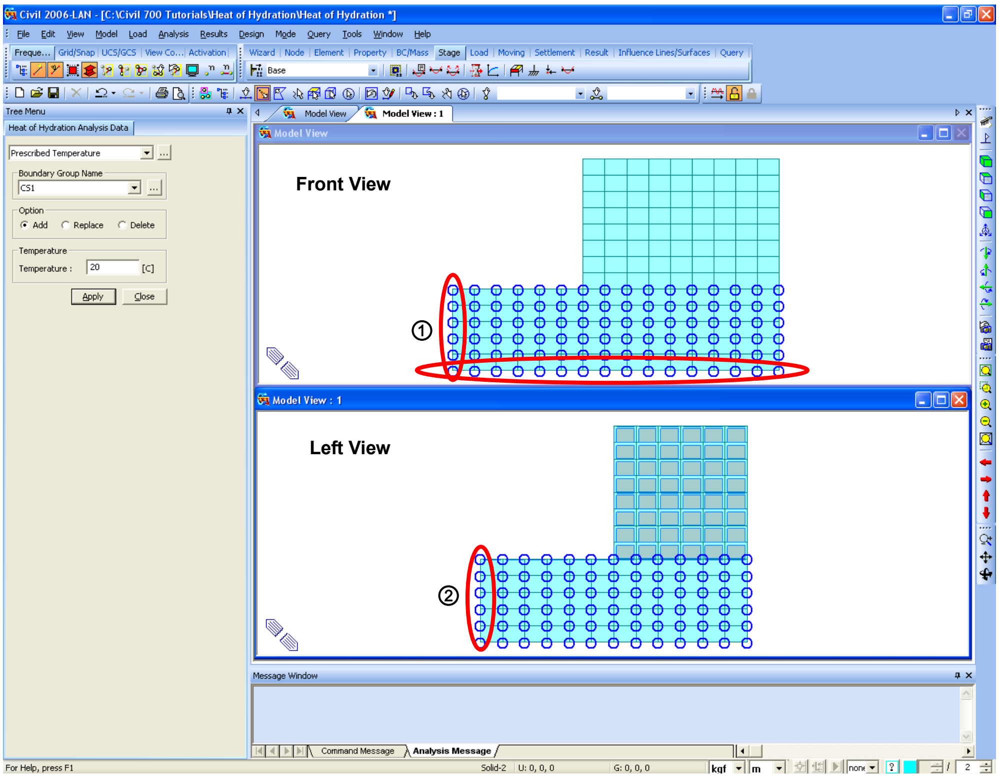


Figure 28 Inputting a constant temperature condition

Defining heat source functions

Heat source functions define the state of emitting heat in the process of hydration, which are dependent on the type of cement and unit cement content. For commonly used concrete mix design, maximum adiabatic temperature rise and reactive velocity coefficient are automatically calculated based on experimental equations and entered if the cement type, casting temperature and unit cement content are specified.

Load / Heat of Hydration Analysis Data / ***Heat Source Functions***

Function Name>(**Heat Source Function**)

Function Type>**Code**

Function>Maximize adiabatic temp. rise (K)>**(33.97)**

Reactive velocity coefficient (a)>**(0.605)** ; Redraw Graph ↵

Function>Maximize adiabatic temp. rise (K)

Reactive velocity coefficient (a)

This example assumes that low hydration heat cement is used, and experimental values of 'K' & 'a' are considered.

Refer to "Heat of Hydration Analysis" in the Analysis Manual.

If experimental value for the maximum adiabatic temperature rise for concrete is available, the "User" Function Type can be used. The "User" Function Type can be inputted either by Heat Source or by Temperature method.

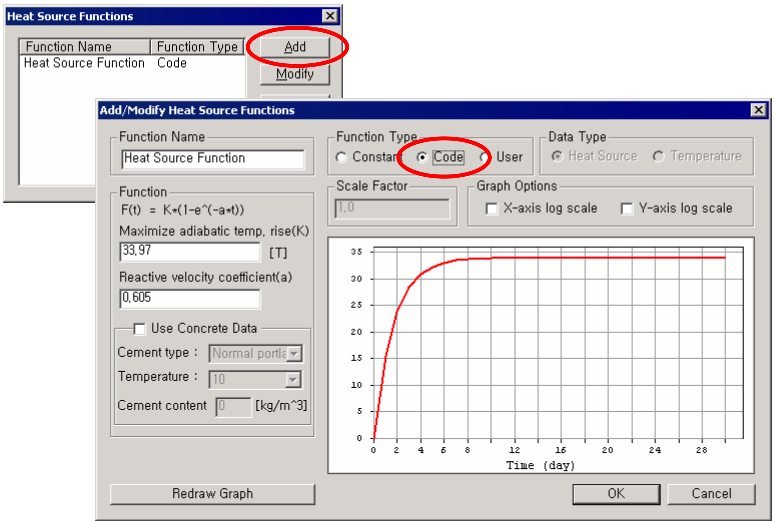


Figure 29 Defining heat source function

Assign the defined heat source function to the concrete.

Load / Heat of Hydration Analysis Data / ***Assign Heat Source***

Select Window (① in **Figure 30**)

Option>**Add/Replace**

Heat Source>**Heat Source Function** ↵

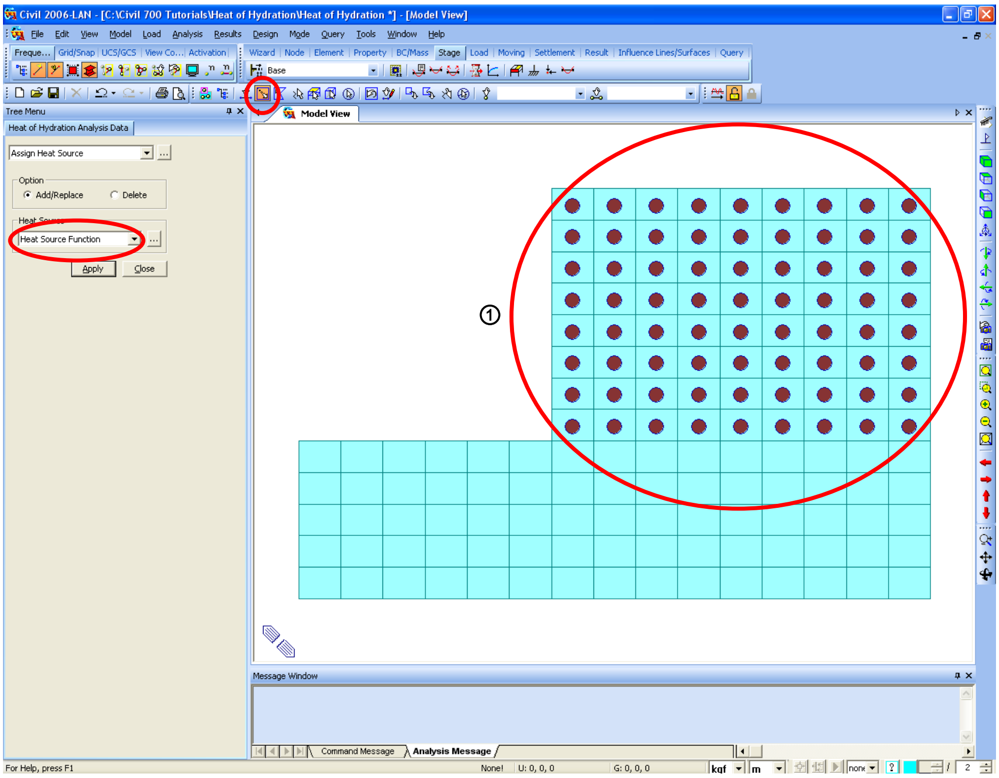


Figure 30 Assigning heat source function

Defining construction stages

Using the previously defined Structure Groups, Boundary Groups and Load Group, we will now specify times for heat of hydration analysis and initial temperature. We will first define the construction stage CS1 for the stage of 1st concrete pour.

Load / Heat of Hydration Analysis Data / ***Define Construction Stage for Hydration***

Stage> Add> Name>**(CS1)**

Initial Temperature>**(20)**

Step>Time(hr)>**(10 20 30 50 80 120 170)** Add

Times inputted in Step are accumulative, not incremental.

Element>Group List>**Subsoil ; Mat Foundation (Lower part)**

Activation> Add

Boundary>Group List>**CS1 ; CS1-Boundary Surface**

Activation> Add ↵

Load>Group List>**Self**

Activation> Add ↵

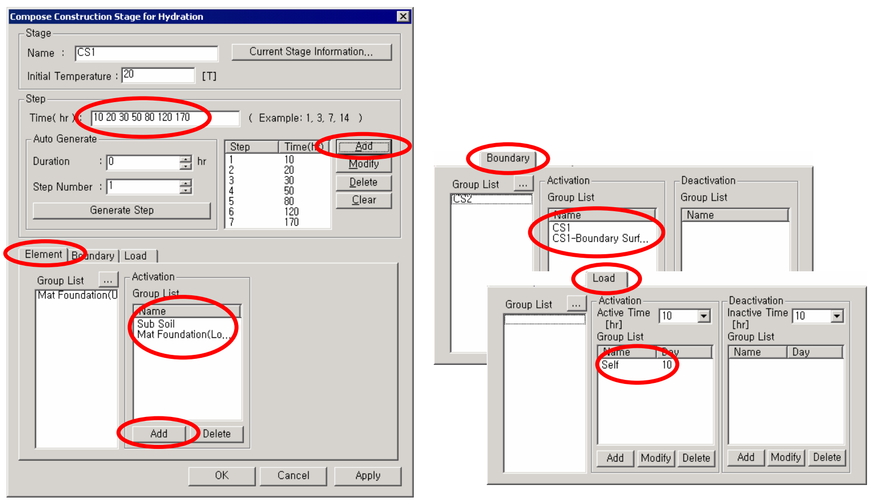


Figure 31 Defining the stage for 1st concrete pour

We then define the construction stage CS2 for the 2nd concrete pour. The duration for the heat of hydration analysis will be 930 hours after the 2nd pour.

Load / Heat of Hydration Analysis Data / ***Define Construction Stage for Hydration***

Stage> Name>**(CS2)**

Initial Temperature>**(19)**

Step>Time(hr)>**(10 20 30 50 80 120 170 300 400 500 600 750 930)** Add

Element>Group List>**Mat Foundation (Upper part)**

Activation> Add

Boundary>Group List> **CS2**

Activation> Add

Boundary>Group List > **CS1-Boundary Surface**

Deactivation> Add

Define the initial temperature for the elements that are activated at the corresponding stage.

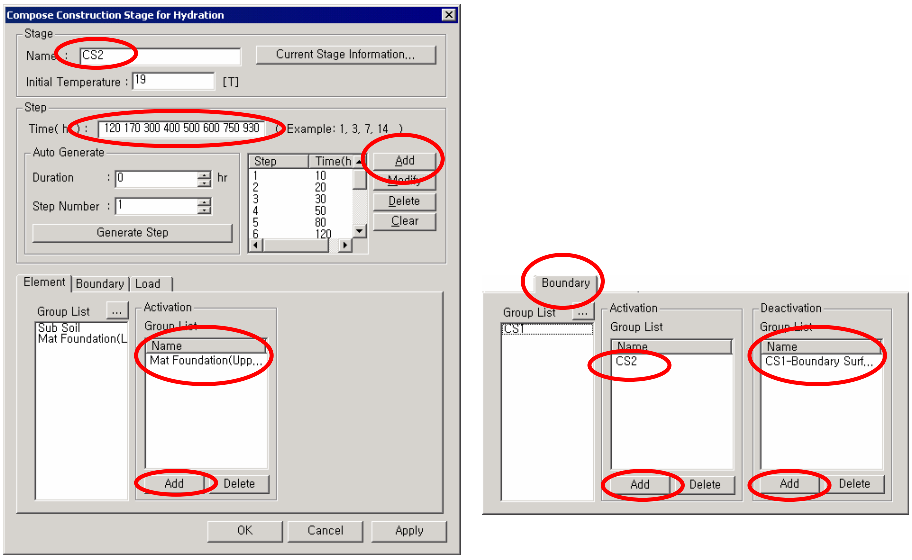


Figure 32 Defining element and boundary groups for the 2nd pour stage

From the Model View, we can check if the Construction Stages are properly defined.

User can either select a stage on the Stage Toolbar or use the keyboard arrows to toggle between different stages while the Toolbar is activated.

Stage>**CS1**

Display

Misc tab

Element Convection Boundary of Heat of Hydration (on) ;

Prescribed Temperature of Heat of Hydration (on) ;

Heat Source for Heat of Hydration (on) ↵

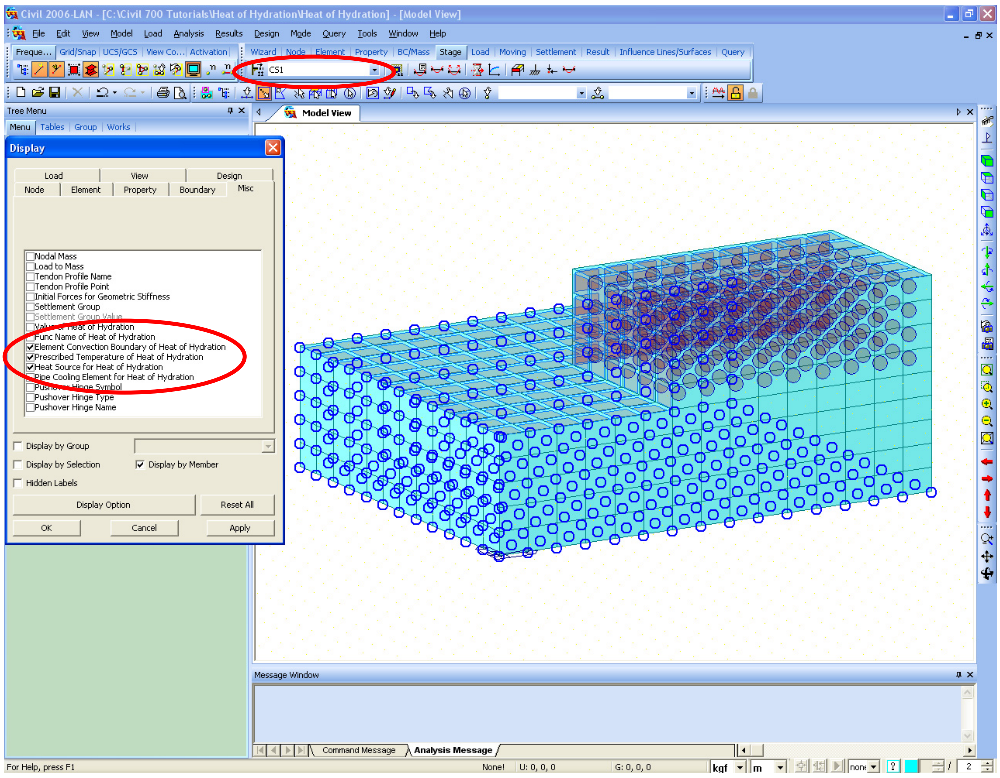


Figure 33 Checking the defined construction stages on the Model View (Stage for the 1st concrete pour)

Structural analysis

We have thus far completed a construction stage model for heat of hydration analysis. We can begin the analysis.

Analysis / ***Perform Analysis***

Analysis results

In this example, the major cause for thermal stresses is due to the temperature differences within the concrete mass resulting in internal constraints. Recapping the overview, Internal Constraints are caused by unequal volume changes. Initially, cooling surface and warm inner parts cause tension at the surface and compression at the inner parts. At a later stage, after the rise in temperature due to heat of hydration reaches the peak level, the cooling (contracting) inner parts relative to the surface cause tension in the inner parts and compression at the surface. The magnitude of the stresses is proportional to the temperature differences between the inner parts and surface. It is also anticipated that the two concrete masses of two separate pours of different ages will exhibit different heat transfer characteristics.

We will analyze the characteristics of thermal stresses in concrete by reviewing the results of heat of hydration analysis reflecting construction stages by graphics, tables, graphs, animations, etc.

Checking change in temperatures

We will check temperature distribution at each step of the construction stages based on the heat of hydration analysis. Figure 34 shows the maximum temperature distribution at the stage of the 1st concrete pour.

Rotate Dynamic (adjust the model view point so that the boundary planes of symmetry can be seen as shown in Figure 34. – Ctrl+Mouse wheel can be also used)

Result / Heat of Hydration Analysis / ***Temperature***

Stage Toolbar>**CS1**

Step>**HY Step 6, 120 Hr**

Type of Display>**Contour** (on) ; **Legend** (on) ↵

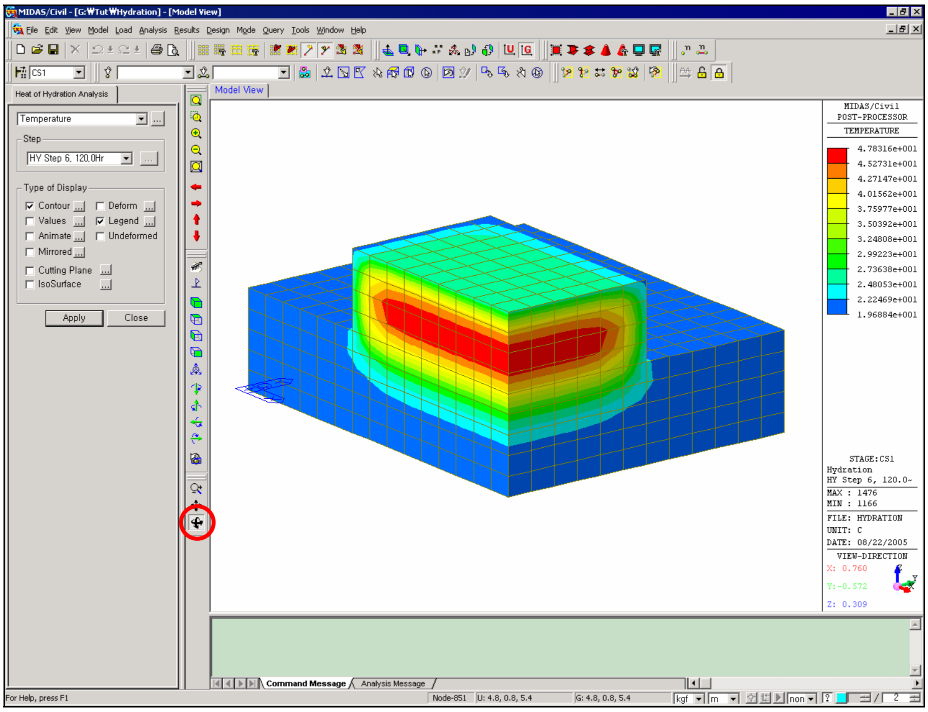


Figure 34 Temperature distribution (1st pour stage)

Next, we will check the temperature distribution at the construction stage 2. The fact that the analysis accounted for construction stages, we note in Figure 35 that heat source action progresses in the lower part of the mat foundation, which was already cast.

Stage Toolbar>**CS2**

Result / Heat of Hydration Analysis / ***Temperature***

Step>**HY Step 4, 220 Hr**

Type of Display>**Contour** (on) ; **Legend** (on) ↵

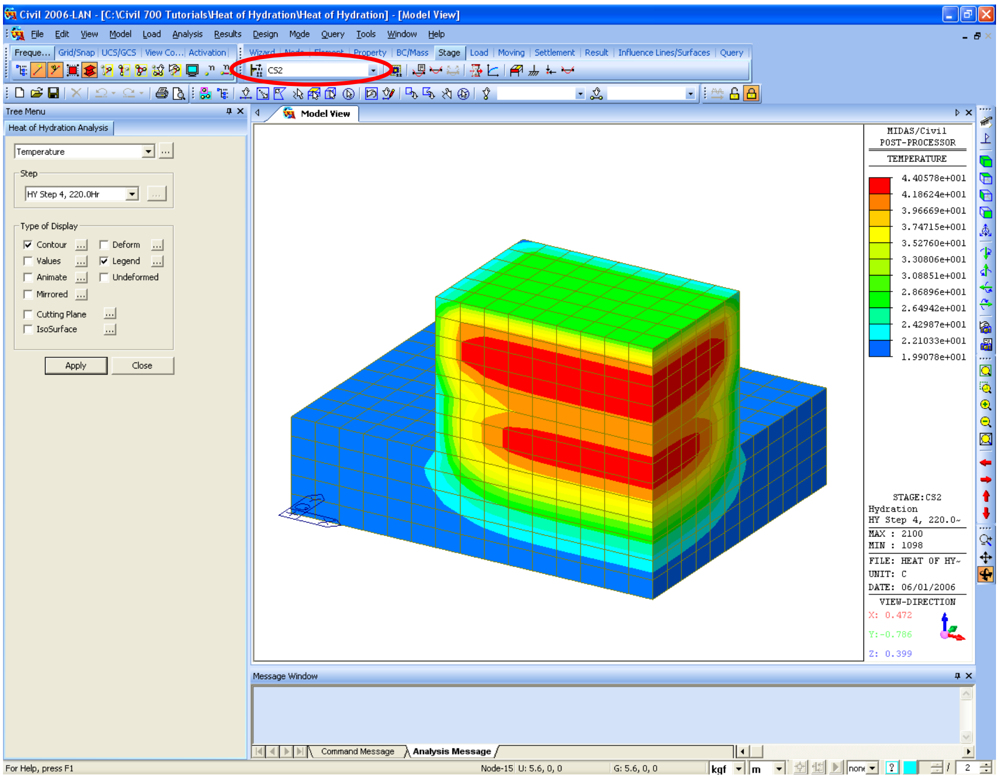


Figure 35 Temperature distribution (2nd pour stage)

Checking change in stresses

We will check the stress distribution of the 1st concrete pour. Figure 36 depicts the stress distribution at which the maximum tension stress occurs on the surface. We will change the unit system to kgf & cm to check stresses.

Status Bar> **kgf** ; **cm**
Stage Toolbar>**CS1**
Result / Heat of Hydration Analysis / ***Stress***
Step>**HY Step 6, 120 Hr**
Stress Option>**Global** ; **Avg.Nodal**
Components>**Sig-XX**
Type of Display>**Contour** (on) ; **Legend** (on) ↵

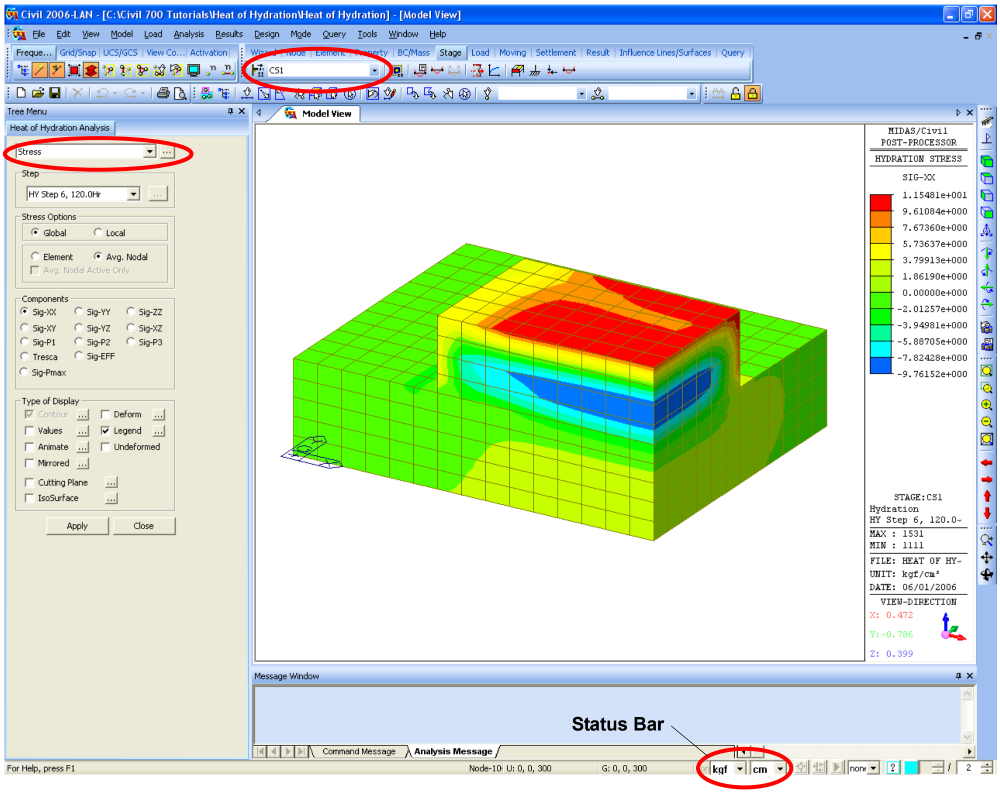


Figure 36 Stress distribution (1st pour stage)

We will check the stress distribution at the 2nd pour stage. As shown in Figure 37, the boundary surface of the first pour shows tension stresses at the early stage of the 2nd pour. The tension stresses at the boundary surface are caused by the increase in volume due to increased temperature in the 2nd pour. This exerts tension on the previously cast concrete.

Stage Toolbar>**CS2**

Result / Heat of Hydration Analysis / ***Stress***

Step>**HY Step 4, 220 Hr**

Stress Option>**Global** ; **Avg.Nodal**

Components>**Sig-XX**

Type of Display>**Contour** (on) ; **Legend** (on) ↵

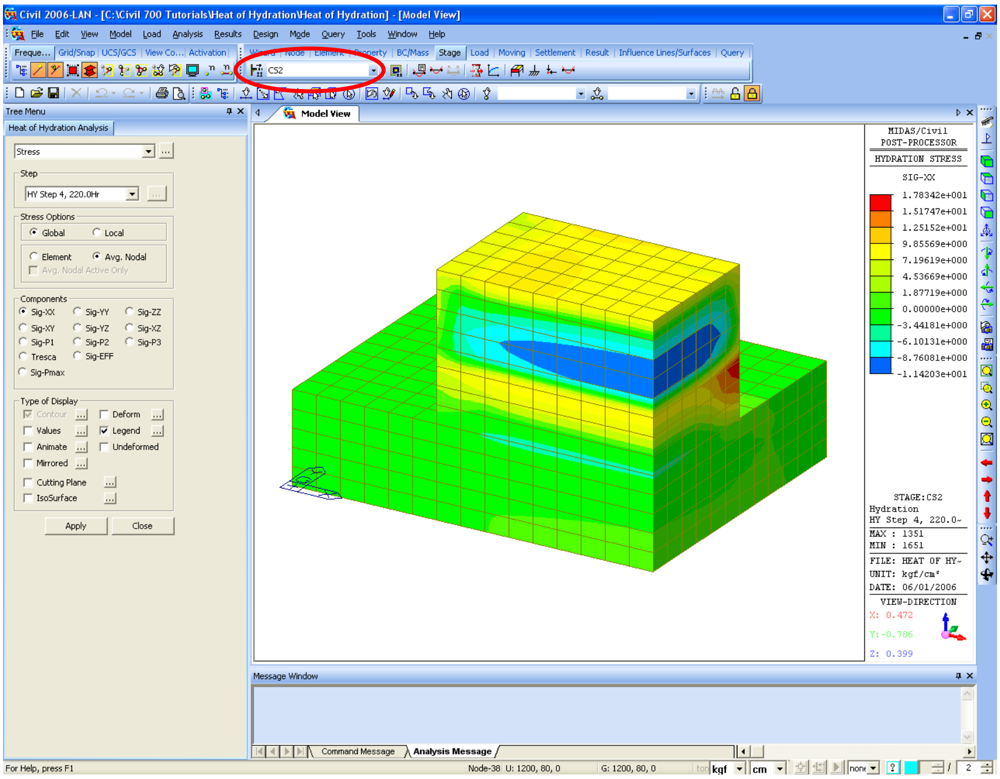


Figure 37 Stress distribution (2nd pour stage)

Checking time history graphs

We will check the graphical results of heat of hydration analysis at various construction stages for specific points. Generally, a user checks the parts where maximum tension stresses are anticipated. In this example, we will select a few points simply based on convenience to sufficiently demonstrate the trend of the analysis results as shown in Figure 38. We will first assign the nodes for generating results.

1st pour concrete: Interior (1476), Surface (1988)
2nd pour concrete: Interior (2308), Surface (2818)

Result / Heat of Hydration Analysis / ***Graph***

Add >Node Define>Node **(1476)** ; Stress Components> **Sig-XX** ↵
Add >Node Define>Node **(1988)** ; Stress Components> **Sig-XX** ↵
Add >Node Define>Node **(2308)** ; Stress Components> **Sig-XX** ↵
Add >Node Define>Node **(2818)** ; Stress Components> **Sig- XX** ↵

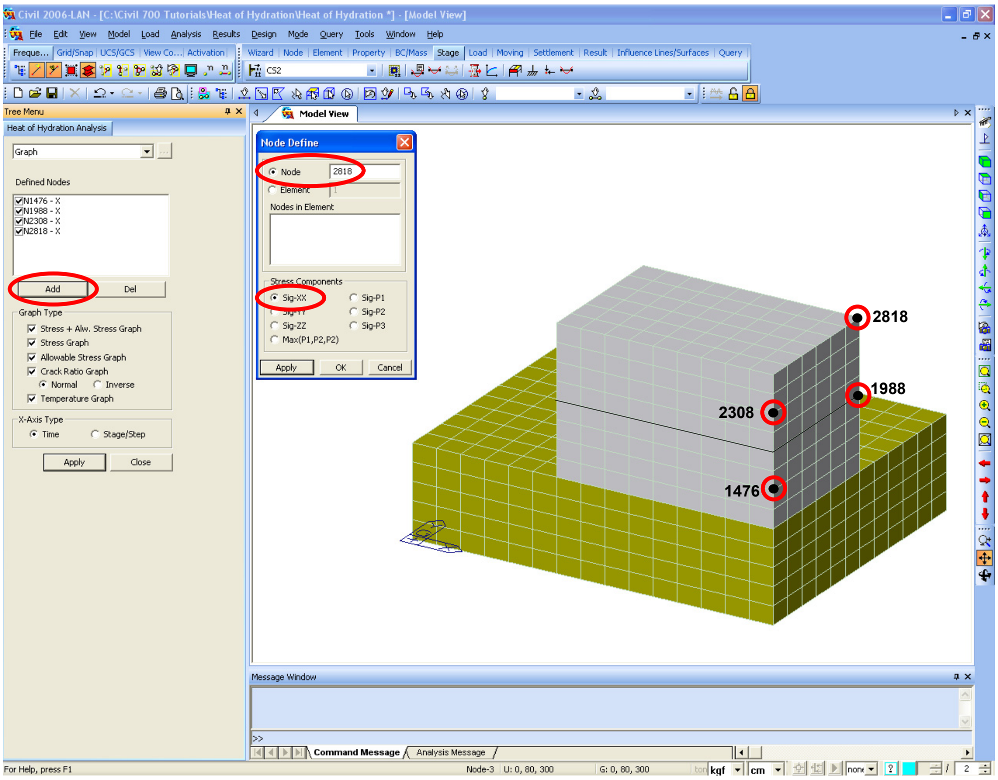


Figure 38 Defining nodes for generating graphs

The time history graph for an interior point (node: 1476) during the 1st pour is shown below.

Result / Heat of Hydration Analysis / ***Graph***

Defined Nodes>**N1476-X**(on)

Graph Type> **Stress + Alw. Stress Graph** (on) ; **Temperature Graph** (on)

Crack Ratio Graph (on) → **Normal** (on)

X-Axis Type>**Time** ↵

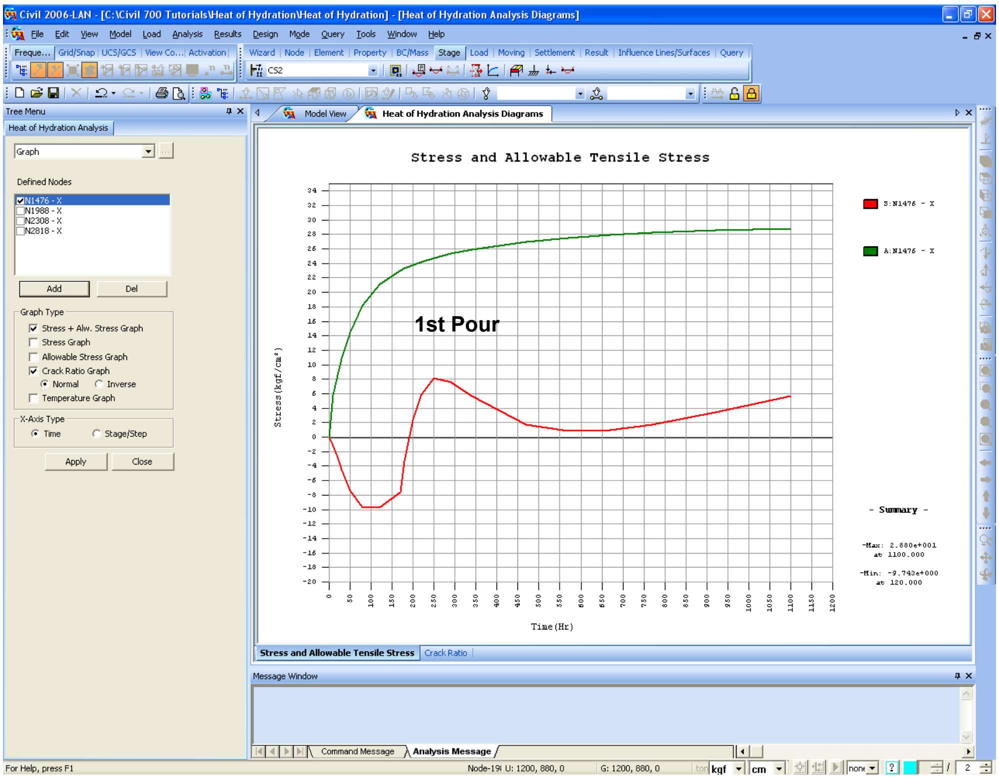


Figure 39 Time history graph of stresses at an interior point of the 1st pour

Next, we will review the results of time history of a point (node: 1988) on the construction joint surface between the 1st and 2nd pours. We will also note that the expansion of the 2nd pour due to temperature rise exerts tension on the 1st pour.

Result / Heat of Hydration Analysis / ***Graph***

Defined Nodes>**N1988-X** (on)

Graph Type>**Stress + Alw. Stress Graph** (on)

Temperature Graph (on)

X-Axis Type>**Time** ↵

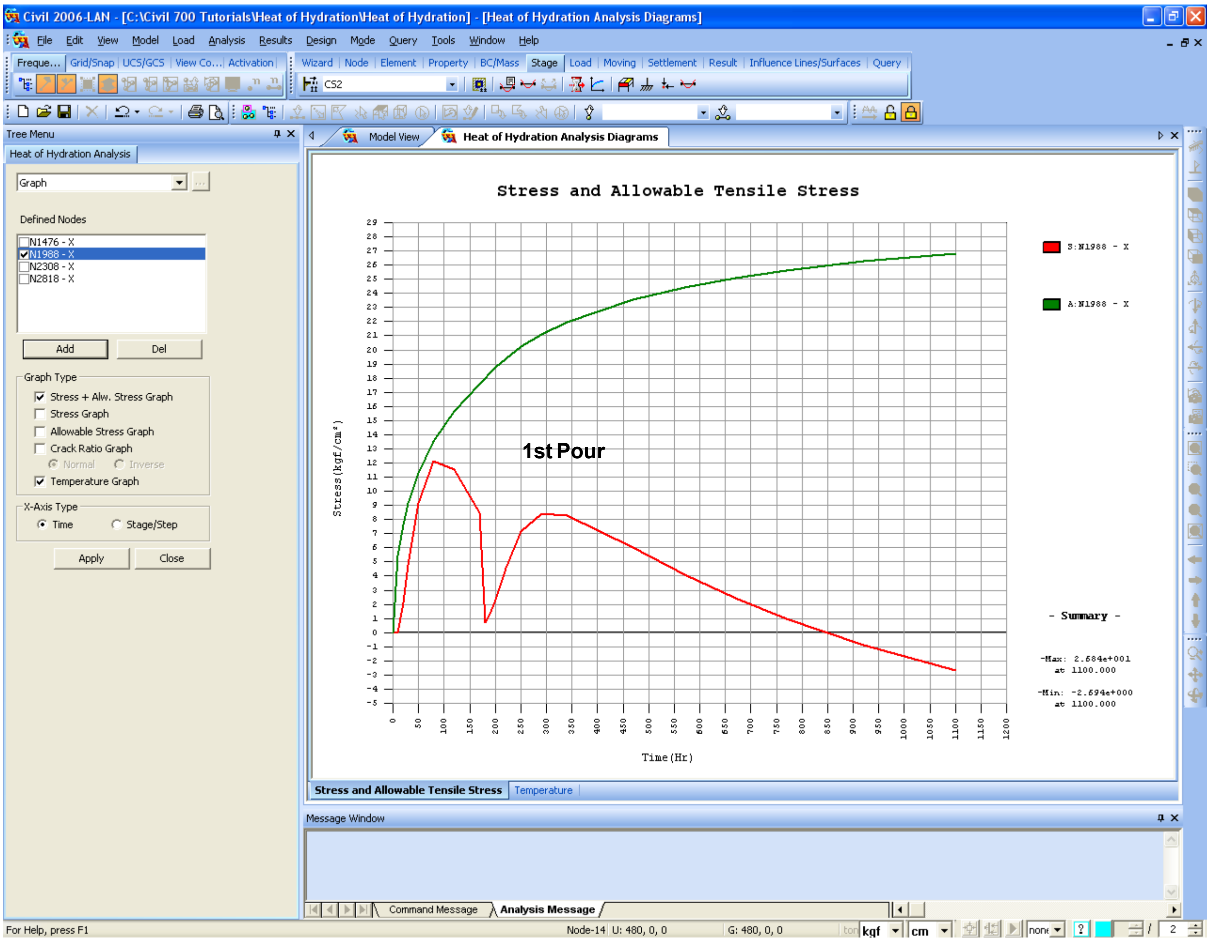


Figure 40 Time history graph of stresses at a surface point of the 1st pour

We will finally check the temperature time history of the interior and surface points during the 1st pour.

Result / Hear of Hydration Analysis / ***Graph***

Defined Nodes>**N1476-X**(on) ; **N1988-X**(on)

Graph Type> **Temperature Graph** (on)

X-Axis Type>**Time** ↵

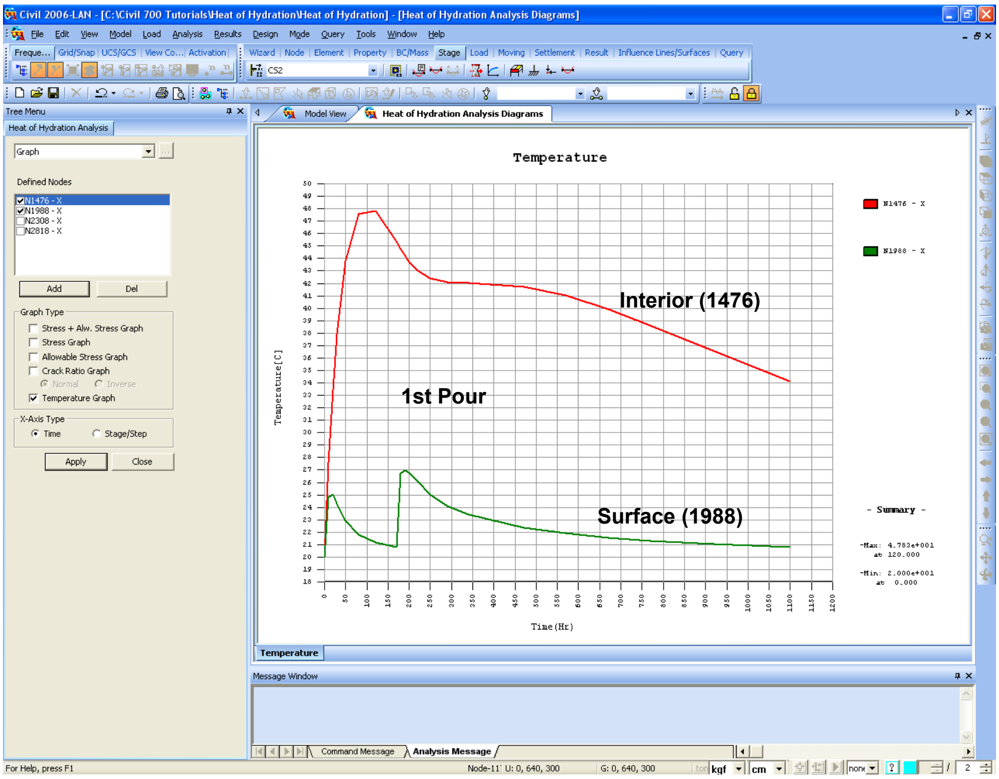


Figure 41 Temperature history graphs of interior and surface points of the 1st pour

Checking results in animation

Finally, we will review the change in temperature (or stress) by construction stages by animation.

Result / Heat of Hydration Analysis / ***Temperature***

Type of Display>**Contour** (on) ; **Legend** (on) ; **Animate**

Animation Details>**Animate Contour** (on) ; **Repeat Full Cycle**

Construction Stage Option>**Stage Animation**>From>**CS1** ; To>**CS2** ↵

Record ↵

Close

In order to save the animation in a file, click the ***Save*** button while the animation is in progress, upon which it is saved as an *.avi* file.

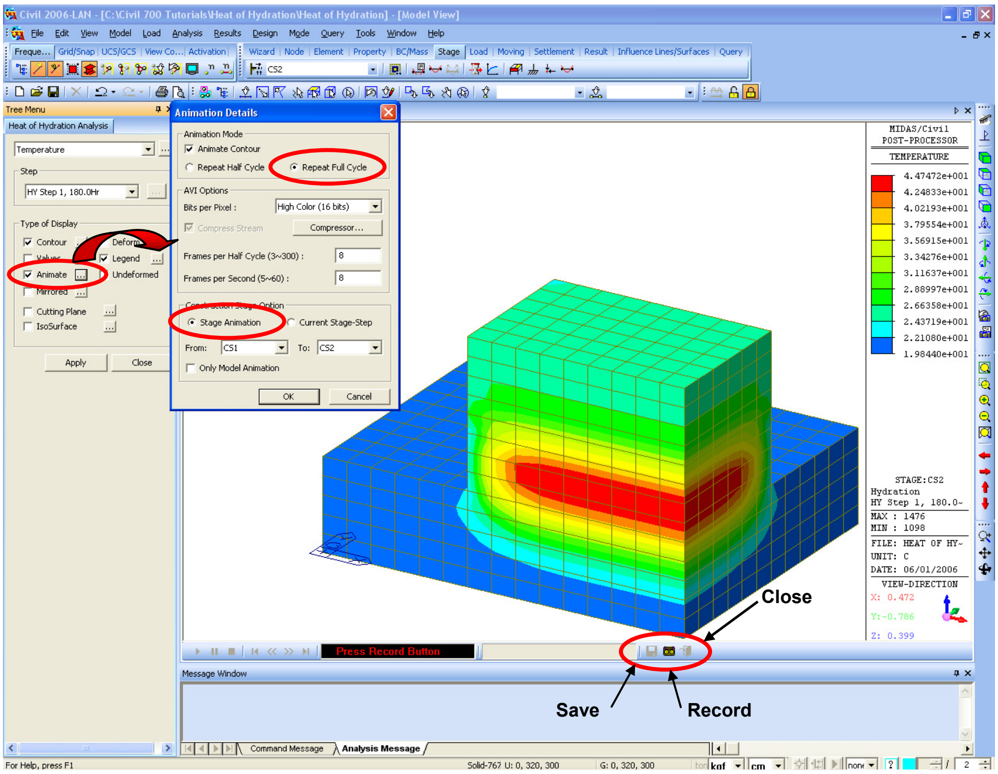


Figure 42 Checking change in temperature by animation