Research on modelling of intake tower in three-dimension CAD software and simulation analysis in FE software

Hongyang Zhang*

North China University of Water Resources and Electric Power, Zhengzhou, China

Abstract

ANSYS is one of the most influential finite element analysis software in the world because of its very powerful calculation and analysis ability, but its pre-process function is weak relatively. SolidWorks is the three-dimension parametric feature modelling software of 100% feature modelling and 100% parameterization, which provides product-level automated design tools. In this paper, combining with the intake tower, it discusses the method of modelling in three-dimension CAD software SolidWorks and the interface processing between SolidWorks and the ANSYS code, which decreases the difficulty in modelling complicated models in ANSYS. In view of the function of the birth-death, element and secondary development with APDL (ANSYS parametric design language), simulation analyses of thermal field and stress during the construction and impounding periods were conveniently conducted.

Keywords: Modelling in SolidWorks, interface processing, birth-death element, APDL, simulation analysis in ANSYS

1 Introduction

Recently, mass concrete is widely applied to hydraulic engineering. The internal temperature of concrete rises due to cement hydration heat, and thus generates thermal stress of the concrete structure. Excessive stress may cause concrete cracks, which affects safety of the concrete structure. It is necessary to analyse the thermal field and thermal stress of important mass concrete structures with both routine methods and the finite element method (FEM). Some researchers have done a large amount of simulation analyses using FE software [1-6], but difficulties in these methods remain. There are two main difficulties:

(i) Modelling of mass concrete structures in FEM software are difficult because of their complexity [7-8].

(ii) The construction processes and boundary conditions of concrete structure are complex, so complete simulation is difficult with FEM software [9-10].

How to solve these two difficulties is of concern to engineers and researchers.

SolidWorks is a CAD/CAE/CAM/PDM desktop system, and the first three-dimension mechanical CAD software in Windows developed by the SolidWorks Company. It provides product-level automated design tools [11]. Since its introduction in 1995, SolidWorks has become a favourite design tool for many of today's engineers, mechanical designers, and industrial designers. In part because of its easy-to-learn graphical user interface and powerful set of tools, SolidWorks is used by many top companies worldwide to design, engineer, and document their products in a variety of fields. At the core

of SolidWorks is the ability to create parametric three-dimension solid geometry that is then used to create drawings, manufacturing instructions, instruction manuals, animations, full-colour renderings, and other types of documentation. Regardless of the complexity of the item being created, the creation process is easy and follows the same basic steps. Firstly, a sketch is created that is turned into a base feature. The base feature is then further refined by adding features that add or remove material from the base feature. Individual part models can then be used to build assemblies that represent the final design. After creating the three-dimension part or assembly models, drawings are made to document the design and manufacturing process. SolidWorks is the three-dimension parametric feature modelling software of 100% feature modelling and 100% parameterization, which has the following outstanding characteristics.

(i) Characteristics of the administrator functions.

(ii) Full-related data management of the zero part design, assembly design and two-dimension drawings, which are interrelated.

(iii) With tightly interface of a number of CAM, CAE software.

At present, modelling by SolidWorks is mainly applied to railway, aerospace, machinery manufacturing, national defense industry, electronics, shipbuilding and other fields [12-15], and is rarely applied to hydraulic engineering.

ANSYS is a type of large universal finite element software that has a powerful ability to calculate and analyse aspects of structure, thermal properties, fluid, electromagnetic, acoustics and so on, which has been

Corresponding author e-mail: 1694440@qq.com

widely used in civil engineering, traffic, water conservancy, railway, petroleum chemical industry, aerospace, machinery manufacturing, national defense, electronics, shipbuilding, biological medicine, geological mining, household appliances and other general industrial and scientific research [16]. Since developed in 1970 by John Dr Swanson at the university of Pittsburgh, ANSYS has occupied a pivotal position in the FES field and been widely accepted by the industrial areas, which is recognized as the standard analysis software of more than 20 professional and technical associations. But the pre-process function of ANSYS is weak relatively. The modelling ability in ANSYS is inferior and the complex model building is very complex by ANSYS. So the pre-processing wastes most time and affects work efficiency seriously.

In the paper, the structure of intake tower was modelled in the three-dimension CAD software SolidWorks and imported into ANSYS with an interface tool. Then, the simulation analysis during construction and impounding periods was conducted by the APDL program in ANSYS.

2 Modelling in SolidWorks and interface processing between SolidWorks and ANSYS

2.1 MODELLING IN SOLIDWORKS

The process of modelling is undertaken from the inside to the outside because of the complexity of internal structure of the intake tower.

(i) The base plate and the left sidewall modelling. Firstly, selecting sketch map and using sketch-rendering tools to draw L-shaped cross-section of base plate and sidewall. Secondly, entering the values of stretching length and stretching the cross-section by stretching convex body command, and the three-dimension model of base plate and the left sidewall is obtained, which is shown in Figure 1.

(ii) Internal structure modelling. Firstly, selecting sketch map and drawing the two-dimension plan for stretching. Secondly, entering the values of stretching length and stretching the plan by stretching convex body command, and the three-dimension model internal structure is obtained. The internal structure model is shown in Figure2.
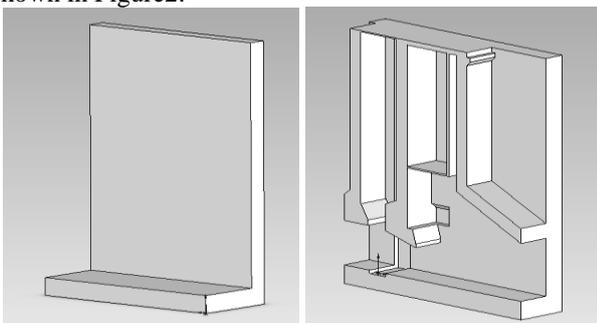

FIGURE 1 3-d L-shaped model   FIGURE 2 Internal structure model

(iii) Optimal model. Modelling the right side wall and modifying the model by stretching removal command. The optimal model is shown in Figure 3. In addition, SolidWorks provides the command of insert→ characteristics→split for model splitting, by which the simulation analyses for the construction impounding of the mass structure become convenient.

(iv) Examining the section of model from different perspectives. In SolidWorks, we can easily check the front and back, up and down, left and right sides of model, and even different location of the section, which helps us conveniently check whether the model is optimal. The cross section of model is shown in Figure 4.
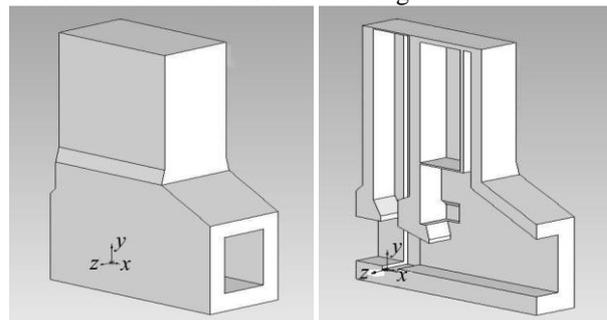

FIGURE 3 Integrated model   FIGURE 4 Cross section

2.2 INTERFACES BETWEEN SOLIDWORKS AND ANSYS

ANSYS provides some interface tools for CAD software, which imports the CAD model conveniently and reduces the difficulties of model processing. The interface tools are given in Table 1.

TABLE 1 CAD software packages and preferred interface tools

CAD software package	File type	Interface tool
AutoCAD	*.sat	Interface tool for SAT
Pro/ENGINEER	*.prt	Interface tool for Pro/ENGINEER
SolidWorks	*.x_t	Interface tool for Parasolid

The model needs to save as type Parasolid (*.x_t) to import into ANSYS correctly, concrete steps are as follows, choose "file→ save as", it is requested to write the model's name and choose the saving type as Parasolid (*.x_t) in the ejecting dialog box, then choose the saving folder and save. In ANSYS, using the command "PARAIN, Name, Extension, Path, Entity, FMT, Scale" or choosing "File→Import→PARA..." in the GUI interface. There are two means of importing, and the differences of whether selecting "Allow Defeaturing" or not are given in Figure 5 and Figure 6. Both of models can be pre-processed in ANSYS in practice.
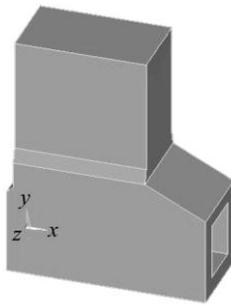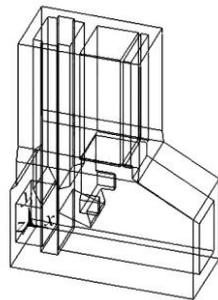
FIGURE 5 With defeaturing   FIGURE 6 Without defeaturing

3 Analysis of thermal field of intake tower

There are 5 products of ANSYS to thermal analysis as shown in Figure 7, which can be classified into ANSYS/ Multiphysics, ANSYS/Mechanical, ANSYS/Thermal, ANSYS/FLOTRAN and ANSYS/ED.
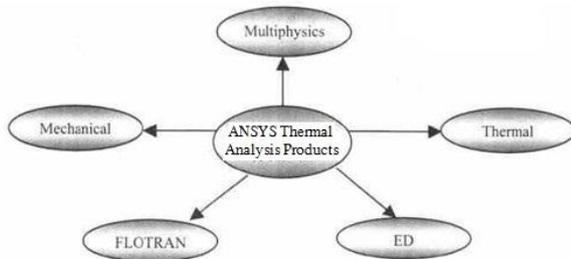

FIGURE 7 ANSYS thermal analysis products

The heat balance equilibrium equation of thermal analysis in ANSYS is based on the principle of energy. In calculating, the temperature of each node is calculated, and then other thermal physical parameters are exported. and export the finite element method to calculate. It can figure out the problems of heat conduction, convection and radiation. In addition, the problems of phase transition, inner heat source and the thermal contact resistance also can be analysed [17-19].

The thermal analysis in ANSYS can be classified into steady heat transfer analysis and transient heat transfer analysis. The steady heat transfer is that the thermal field does not change with time, and the transient heat transfer is that the thermal field changes significantly with time.

The temperature analysis of the intake tower during the construction and impounding periods involves aspects of the thermal field and thermal stress. The calculation must deal with the problems of simulation of layered construction, dynamic boundary conditions, hydration heat, dynamic elasticity modulus, autogenous volume deformation of concrete and thermal creep stress, which are difficult to simulate directly in ANSYS. APDL is a scripting language based on the style of parametric variables. It is used to reduce a large amount of repetitive work in analysis [20-22]. This study carried out a simulation analysis of the thermal field considering nearly all conditions of construction, using the birth-death element and programming with APDL.

Killing the elements is not removing the element from the model, but reducing its stiffness by multiplying a small factor with its stiffness matrix. At the same time, the element loads, strains, mass, damp, specific heat and other similar parameters of the dead element are set to zero, which are not included in the results. As the same process as above, the activated element is not added to the model, but to re-activate it, then the stiffness, mass and other parameters return to the original values. All elements must be generated in PREP7, including to the dead elements which will be activated later, because the elements cannot be generated in equation solver. In analysis, all elements are killed at the beginning and the elements, which are needed to analyse are re-activated later.

The commands of setting birth-death element by APDL are written as,

ESEL, …! Selecting the elements which will be killed
EKILL, …! Killing the elements
ESEL,… !Selecting the elements which will be activated
EALIVE, …! Activating the elements

It can be used the loop commands to realize the layered construction of concrete, which solves the problem of simulating difficultly in GUI interface of ANSYS. The loop commands are written as,

*DO, Par, IVAL, FVAL, INC
Commands section
*ENDDO

The written commands can be saved in text file. Using the command "/INPUT, 'file name', 'txt', 'saving path' " or choosing "File→read input from..." in the GUI interface, and importing the text file with commands, which realize the simulation of construction progress conveniently.

The life and death of element functions are realized by modifying the stiffness. When the element was "killed", the stiffness matrix of which is not removed, but its values reduced to a minimum. The stiffness of killed elements multiplied by a tiny reduction factor (default is 1e-6), and the values of the stiffness is not zero to prevent matrix singularity.

3.1 ANALYSIS OF THERMAL FIELD OF INTAKE TOWER
3.1.1 Unsteady thermal field analysis

Many factors such as the effect of hydration heat of cement, air temperature and water temperature cause the temperature of concrete changes. This is a heat conduction problem of internal heat sources in the area. The unsteady thermal field is written as [23]:

$$\frac{\partial T}{\partial \tau} = \frac{\lambda}{c\rho}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) + \frac{\partial \theta}{\partial \tau} \qquad , \qquad (1)$$

where τ is the age of concrete, c is the specific heat of concrete, ρ is the density of concrete, λ is the thermal conductivity of concrete, and θ is the adiabatic temperature rise of concrete.

For the three-dimension unsteady thermal field, the functional form $I^e(T)$ is:

$$I^e(T) = \iiint_{\Delta R} \left\{ \frac{1}{2}\alpha \left[\left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2 + \left(\frac{\partial T}{\partial z}\right)^2 \right] + \right.$$
$$\left. \iiint_{\Delta R} \left(\frac{\partial T}{\partial \tau} - \frac{\partial \theta}{\partial \tau} \right) T \right\} dxdydz + \iint_{\Delta D} \bar{\beta} \left(\frac{1}{2}T^2 - T_a T \right) ds \quad (2)$$

where ΔR is a subfield of unit e, the thermal diffusivity $\alpha = \frac{\lambda}{c\rho}$, $\bar{\beta} = \frac{\beta}{c\rho}$, β is the exothermic coefficient, ΔD is the area on surface D, which is only in boundary units, and T_a is the air temperature.

3.1.2 Initial and boundary conditions

In analysis, the calculated initial temperature of concrete is 10 ℃.

The index formula of hydration heat of cement is written as [24-25]:

$$Q(t) = 71610 \left[1 - \exp(-0.36t) \right], \quad (3)$$

where t is the pouring time, and Q is the hydration heat. The relation between Q and θ is written as:

$$\frac{\partial \theta}{\partial \tau} = \frac{Q}{c\rho}. \quad (4)$$

The boundary conditions involve the laws of interaction between concrete and the surrounding medium. When concrete is exposed to the water, the boundary condition is:

$$T(\tau) = f(\tau). \quad (5)$$

When concrete is exposed to the air, the boundary condition is:

$$-\lambda \left(\frac{\partial T}{\partial n} \right) = \beta (T - T_a), \quad (6)$$

where n is the normal direction. Both T_a and β are constants or variables.

The steel formworks and straws are used as the insulation materials during the maintenance period, and the exothermic coefficients of which are $45kJ/(m^2 \bullet h \bullet^\circ C)$ and $10kJ/(m^2 \bullet h \bullet^\circ C)$, respectively.

The air temperature variation formula is written as:

$$T = 26.1 - 25.1\cos\left[\frac{\pi}{284}(t-79) \right]. \quad (7)$$

The air temperature formula during impounding period is written as:

$$T = 15.8 + 11.9 \times \cos[\pi \times (t-281)/180]. \quad (8)$$

After impounding finished, the following formulas are given to describe the water temperature variation of different height:
Height from 89m to 107m, $T = 18^\circ C$;
Height from 107m to 110m;

$$T = 18 + 2 \times \cos[\pi \times (t-281)/180]. \quad (9)$$

Height from 110m to 112m;

$$T = 18 + 3 \times \cos[\pi \times (t-281)/180]. \quad (10)$$

Height from 112m to 113m,

$$T = 18 + 5 \times \cos[\pi \times (t-281)/180]. \quad (11)$$

3.2 THERMAL FIELD ANALYSIS IN ANSYS

Table 2 shows the construction scheme of layered construction. A layer is not poured until the former layer is poured. The pouring days in Table 2 are all the total days of construction for each layer.

TABLE 2 Construction scheme

Construction elevation (m)	Pouring day (d)	Construction elevation (m)	Pouring day (d)
86.5-89.0	1-25	103.0-108.0	122-152
89.0-95.0	26-50	108.0-114.0	153-179
95.0-96.8	51-89	114.0-120.0	180-201
96.8-103.0	90-121	120.0-121.0	202-221

In analysis, the simulation of layered construction and impounding is settled conveniently by function of the birth-death element and secondary development with APDL (ANSYS parametric design language).

The coordinates and maximum temperatures of feature points in every layer are given in Table 3, and the temperature curves are shown in Figure 8.

TABLE 3 Coordinates and maximum temperature of feature points

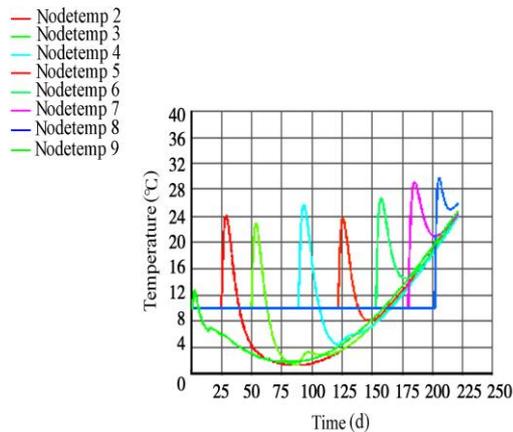| Feature point number	x	y	z	height	Maximum temperature (ºC)
2	7.4	6.0	1.0	92.5	24.268
3	8.4	9.9	5.0	96.4	24.353
4	16.4		5.0	102.5	25.743
5	8.4	20.0	5.0	106.5	24.346
6	16.4	25.0	8.0	111.5	26.611
7	1.0	30.0	2.0	116.5	29.224
8	8.4	34.5	5.0	121.0	29.934

FIGURE 8 Maximum temperature curves

In Figure 8, the curves from Nodetemp 2 to Nodetemp 8 express temperature variation of feature points from 2 to 8, and the curve of Nodetemp 9 is the air temperature curve. It is shown that the maximum temperature occurs on the 3rd or 4th day after pouring and decreases with time. Feature point 4, the coordinates of which are (16.4, 16.0, 5.0), shows the maximum temperature difference of 23.534℃. Feature point 8 shows the maximum temperature rise during the construction period, and the maximum temperature of which is 29.934 ℃, occurring on the 206th day of the total construction period.

The impounding commenced immediately after the construction of the intake tower, which lasted for 16 days, and the calculating lasted for 170 days. In particular, the reservoir elevation started from 84.0m and finished at 108.0m, at the rate of approximately 1.5m per day. The feature points are selected in every layer above the base plate. The maximum temperatures and the temperature curves are given in Table 4 and Figure 9, respectively.

Figure 9 shows that the maximum temperature of each layer occurs on the 60th day after impounding, and then the temperature decreases with time. In Figure 6, the numbers of feature points from 2 to 7 are corresponding to their maximum temperature curves from Nodetemp 2 to Nodetemp 7, and the curve of Nodetemp 8 is the air temperature curve. Feature point 6, the coordinates of which are (5.0, 26.0, 24.0), the maximum temperature of

which underwater is 24.413 ℃, occurring on the 285th day of the total construction and impounding periods.

TABLE 4 Coordinates and maximum temperature of feature points

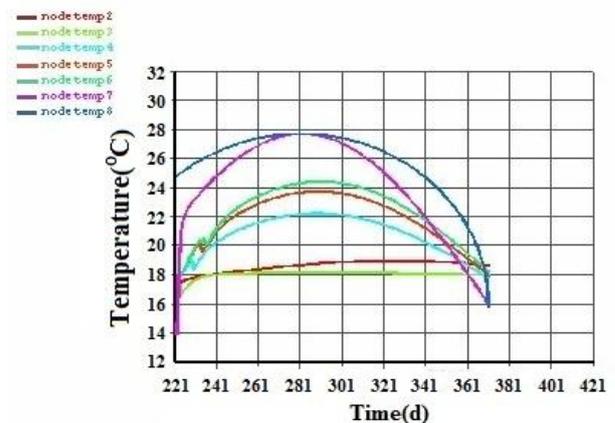| Feature point number	x	y	z	height	Maximum temperature (℃)
2	5.0	2.0	23.0	88.5	18.979
3	1.0	7.0	23.0	93.5	18.145
4	10.0	15.04	7.632	101.54	22.221
5	5.0	24.0	8.632	110.5	23.729
6	5.0	26.0	24.0	112.5	24.413
7	5.0	34.5	17.632	121.0	27.698
8			Air temperature variation curve		

FIGURE 9 Maximum temperature curves

4 Analysis of thermal stress of intake tower

Expansion or contraction of the structure occurs along with heating and cooling. The thermal stress occurs when the expansion or contraction is limited. In this paper, the temperature of nodes was applied to the structure as a body load after the analysis of the thermal field.

4.1 SELECTION OF CALCULATING PARAMETERS

The parameters of concrete are given in Table 5.

TABLE 5 Parameters of concrete

Material	Density (kg/m³)	Coefficient of linear expansion (1/℃)	Poisson ratio
Concrete	2 447.5	9×10^{-6}	0.167

The elasticity modulus is written as:

$$E_1 = 3.6 \times 10^{10} \left[1 - \exp\left(-0.40 t^{0.34} \right) \right]. \tag{12}$$

The creep effect is considered in analysis of temperature stress, and the formula of the creep degree is written as:

$$C = \left[0.23 \left(1 + 9.2t^{-0.45} \right) \left(1 - e^{t_1} \right) + 0.52 \left(1 + 1.17t^{-0.45} \right) \right.$$
$$\left. \times \left(1 - e^{t_2} \right) \right] \times 10^{-10} \big/ 3.60 \qquad .(13)$$

The creep degree is influenced by the cement type, water-cement ratio and admixture, where $t_1 = -0.3(t-3)$ and $t_2 = -0.005(t-3)$. When $t_1 < -80$, we consider $t_1 = -80$; and when $t_2 < -80$, we consider $t_2 = -80$.

Considering the creep degree, the formula of the elasticity modulus is adjusted to be,

$$E = E_1 \big/ \left(1 + CE_1 \right). \qquad (14)$$

4.2 THERMAL STRESS ANALYSIS IN ANSYS

ANSYS software provides the following two methods of thermal stress analysis,

(i) The indirect method. Thermal field analysis is constructed first, and then the node temperature is applied to the structure as a body load.

(ii) The direct method. The results of thermal field and thermal stress are gained by using coupling elements with both temperature and displacement degree of freedom.

In this paper, the thermal field and thermal stress analyses during the construction and impounding periods belongs to the conditions that temperature of nodes are unknown and thermal and structural coupling is unidirectional, so the first method which is the indirect method is selected in analysis.

The coordinates of feature points in thermal stress analysis were same as those in thermal field analysis. Table 6 shows the maximum thermal stress of each point. Feature point 9 is the point with the maximum thermal stress.

TABLE 6 Maximum thermal stress of feature points

Feature point number	Maximum thermal stress (MPa)	Feature point number	Maximum thermal stress (MPa)
2	0.25	6	0.17
3	0.26	7	0.13
4	0.38	8	0.14
5	0.37	9	1.68

The thermal stress curves of feature points are shown in Figure 10.

In Figure 10, the curves from S1_2 to S1_9 express the maximum stress variation of feature points from 2 to 9, and the S1_10 curve is the ultimate tensile stress of concrete. The formula of concrete's ultimate tensile stress is written as,

$$\sigma_t = 0.232 \times 10^6 \left\{ 33.5 \left[1 + 0.2 \times \ln \left(t/28 \right) \right] \right\}^{\frac{2}{3}}. \qquad (15)$$
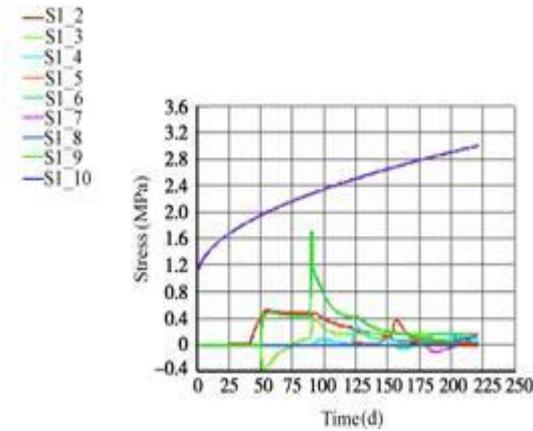

FIGURE 10 Maximum stress curves

The maximum thermal stress occurs on 90th day of the construction period on feature point 9, which is located at the interface between the third layer and the fourth layer, and the value of which is 1.68 MPa. It is known that the thermal stress increases with the temperature difference. Thus, it is postulated that the maximum thermal stress is caused by the instantaneous temperature difference between two layers in the pouring period. It is known that the maximum thermal stress of each point during the construction period is less than the ultimate tensile stress of concrete from Figure 10.

The maximum thermal stress of each point during impounding period is shown in Table 7. Feature point 6, the coordinates of which are (5.0, 26.0, 24.0), is the point with the maximum thermal stress.

TABLE 7 Maximum thermal stress of feature points

Feature point number	Maximum thermal stress (MPa)	Feature point number	Maximum thermal stress (MPa)
2	--	5	0.22
3	0.10	6	0.26
4	0.12	7	0.23

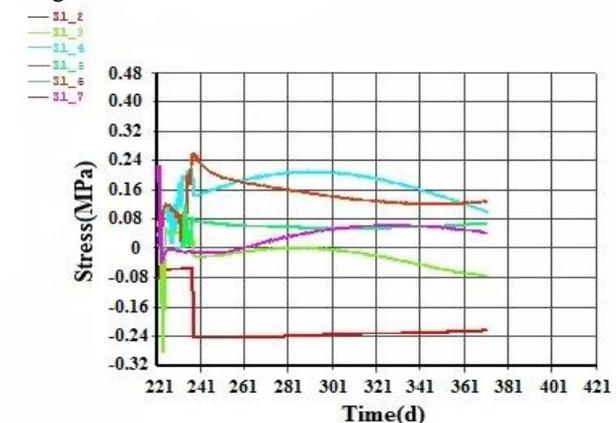The thermal stress curves of feature points are shown in Figure 11.

FIGURE 11 Maximum stress curves

In Figure 11, the numbers of feature points from 2 to 7 are corresponding to their maximum stress curves from S1_2 to S1_7. The figures and table show that the

maximum thermal stress of the intake tower is 0.26 MPa, occurring on the 18th day of the impounding period, and the maximum thermal stress of each point in the intake tower during the impounding period is less than the ultimate tensile stress of concrete.

5 Conclusions

(i) In this paper, the method of modelling in three-dimension CAD software SolidWorks and the interface processing between SolidWorks and the ANSYS code are discussed, which realizes an effective combination of the advantages of both SolidWorks and ANSYS.

(ii) The thermal field and thermal stress during the construction and impounding periods, considering multi factors such as layered construction and impounding, hydration heat, dynamic elasticity modulus, autogenous volume deformation and creep of concrete, are conveniently obtained by birth-death element and secondary development with APDL.

(iii) It is known that the temperature rises rapidly in the early stage of construction, and reaches a maximum value of 29.934 ℃ on the 3rd or 4th day after pouring. The maximum temperature during impounding period is 24.413 ℃, occurring on the early stage of impounding. The thermal stress increases with the temperature difference, and the maximum thermal stress occurs at the interface of new and old layers, which is caused by the instantaneous temperature difference. The values of maximum thermal stress of construction and impounding are 1.68 MPa and 0.26 MPa, which are both less than the ultimate tensile stress of concrete.

Acknowledgments

References

[1] Ashida F, Tauchert T R 1998 *Journal of Applied Mechanics* **65** 367-73

[2] Kawaguchi T, Nakane S 1996 *ACI Materials Journal* **93** 96-101

[3] Lin J Y, Cheng T F 1997 *Numerical Heat Transfer* **32** 187-203

[4] Wu Y, Luna R 2001 *Finite Elements in Analysis and Design* **37** 97-106

[5] Wu J G, Li X J, He K F 2012 *Journal of Computers* **7** 2208-15

[6] Li X J, Wang K, Jiang L L 2012 *Journal of Software* **7** 959-65

[7] Zhang J 2005 *Chinese Quarterly of Mechanics (Shanghai)* **26** 257-62 *(in Chinese)*

[8] Du P, Liu S X, Tan G Z, Liu K X 2012 *Journal of Liaoning Technical University(Natural Science) Fuxin* **31** 526-30 *(in Chinese)*

[9] Zhao Y W, Geng D X, Liu X M 2012 *Journal of Computers* **7** 2742-9

[10] Jin K K, Kook H K, Yang J K 2001 *Computers and Structures* **79** 163-71

[11] Liu L J, Ren J P 2005 *Mechanical Management and Development, (Datong)* **20** 74-75 *(in Chinese)*

[12] Prabhakar S, and Henderson M R 1992 *Computer—Aided Design* **24** 381-93

[13] Rajan G, Venkat D 1998 *Computer-Aided Design* **30** 863-73

[14] Gao Y 2010 *Journal of Yangzhou Polytechnic College* **14** 22-24 *(in Chinese)*

[15] Li X, Huang Z F, Li H T 2013 *Journal of Software* **8** 2517-21

[16] Gao P, Li P H, Bao K M 2012 *Journal of China Three Gorges university (Yichang)* **34** 29-33*(in Chinese)*

[17] Chen G R, Xu W T, Yang Y., Li K 2012 *Chinese Journal of Computational Physics* **29** 411-6 *(in Chinese)*

[18] Li X F 2012 *Concrete* **22** 127-9*(in Chinese)*

[19] Chen S K, Zhang Y P, Guo L X, Wang H. B, Xie Z Q 2012 *Concrete* **22** 10-6 *(in Chinese)*

[20] Liu X H, Ma G., Chang X L, Zhuo W 2012 *Engineering Mechanics* **29** 159-164

[21] Ma C C, Li S. Y, Zhao L J, Liu C Y 2012 *Journal of Northwest AF University(Natural Science)* 40 217-23

[22] Gong S G, Xie G L 2004 *Commands and Parametric Programming in ANSYS* Beijing: China Machine Press *(in Chinese)*

[23] Zhang G X 2004 *Journal of Hydraulic Power (Beijing)* **56** 71-6 *(in Chinese)*

[24] Zhu B F 1994 *Journal of Hydraulic Power (Beijing)* **46** 21-30 *(in Chinese)*

[25] Zhu B F 2006 *Journal of Hydraulic Power (Beijing)* **37** 1424-32 *(in Chinese)*

Author

Hongyang Zhang, born on March 20, 1981,China

Current position, grades: North China University of Water Resources and Electric Power, Hydraulic Structure Engineering, Lecturer
Scientific interest: CAD modelling, model test, and Simulation analysis , Hydraulic Structure Engineering
Experience:

Date	School	Major	Degree
2000-2004,	Zhengzhou university,	Water resources and hydropower engineering,	Bachelor
2004-2007,	Zhengzhou university,	Hydraulic Structure Engineering ,	Master
2007-2010,	Hohai university,	Hydraulic Structure Engineering ,	Doctor
2010-now,	North China University of Water Resources and Electric Power, Hydraulic Structure Engineering, Lecture		