Water Science and Engineering, 2009, 2(1): 95-102
doi:10.3882/j.issn.1674-2370.2009.01.009

http://kkb.hhu.edu.cn
e-mail: wse@hhu.edu.cn

Modeling in SolidWorks and analysis of temperature and thermal stress during construction of intake tower

Hong-yang ZHANG*[1], Tong-chun LI[1], Zong-kun LI[2]

1. College of Water Conservancy and Hydropower Engineering, Hohai University, Nanjing 210098, P. R. China
2. School of Water Conservancy and Environment Engineering, Zhengzhou University, Zhengzhou 450002, P. R. China

Abstract: With a focus on the intake tower of the Yanshan Reservoir, this paper discusses the method of modeling in the 3D CAD software SolidWorks and the interface processing between SolidWorks and the ANSYS code, which decreases the difficulty in modeling complicated models in ANSYS. In view of the function of the birth-death element and secondary development with APDL (ANSYS parametric design language), a simulation analysis of the temperature field and thermal stress during the construction period of the intake tower was conveniently conducted. The results show that the temperature rise is about 29.934 ℃ over 3 or 4 days. The temperature differences between any two points are less than 24 ℃. The thermal stress increases with the temperature difference and reaches its maximum of 1.68 MPa at the interface between two concrete layers.

Key words: *SolidWorks; ANSYS; APDL; birth-death element; temperature field; thermal stress*

1 Introduction

Mass concrete is widely used in civil and hydraulic engineering nowadays, and its thermal stress increasingly attracts attention during design and construction. It is necessary to analyze the temperature 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 FEM software (Tatro 1985; Barrett et al. 1992; Kawaguchi and Nakane 1996; Zhu and Xu 2001; Zhu 2006), but difficulties in these methods remain. There are two main difficulties: (1) Most mass concrete structures are complex and difficult to model with FEM software. (2) Complete simulation is difficult with FEM software because of the complex construction processes and boundary conditions of concrete.

The structure of the intake tower of the Yanshan Reservoir is complex. It is 34.5 m high and there is a square pressure tunnel at the bottom, the side length of which is 6 m. The intake tower was modeled in the 3D CAD software SolidWorks and imported into ANSYS with an interface tool. Then, using the APDL program, analysis of the temperature field and thermal

This work was supported by the Natural Science Foundation of Henan Province (Grant No. 511050100).
*Corresponding author (e-mail: *zhybryant@hhu.edu.cn*)
Received Nov. 11, 2008; accepted Jan. 15, 2009

stress during construction was conducted.

2 Modeling in SolidWorks and interface processing between SolidWorks and ANSYS

2.1 Modeling in SolidWorks

SolidWorks is a CAD/CAE/CAM/PDM desktop system, and the first 3D mechanical CAD software in Windows developed by the SolidWorks company. It provides product-level automated design tools (Liu and Ren 2005).

The outside structure of the intake tower is simple but the internal structure is relatively complex. Therefore, the process of modeling is undertaken from the inside to the outside. The integrated and internal models of the intake tower are shown in Fig. 1 and Fig. 2.

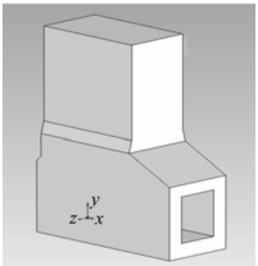

Fig. 1 Integrated model

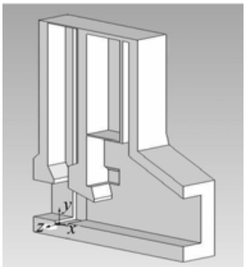

Fig. 2 Cross section

2.2 Interface processing between SolidWorks and ANSYS

ANSYS is a type of large universal finite element software that has a powerful ability to calculate and analyze aspects of structure, thermal properties, fluid, electromagnetics, acoustics and so on. In addition, the interface of ANSYS can be used to import the CAD model conveniently (Zhang 2005), which greatly reduces the difficulties of dealing with complex models. 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

After modeling in SolidWorks, it is necessary to save the model as a type of Parasolid (*.x_t) so as to import it into ANSYS correctly. Then, in ANSYS, the importing of the model is completed with the command “PARAIN, Name, Extension, Path, Entity, FMT, Scale” or the choice of “File→Import→PARA...” in the GUI interface. There are two means of importing: selecting or not selecting “Allow Defeaturing”, the differences of which are shown in Fig. 3 and Fig. 4.

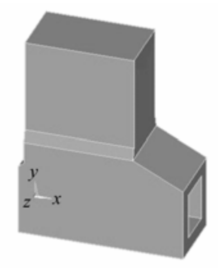

Fig. 3 Importing with defeaturing

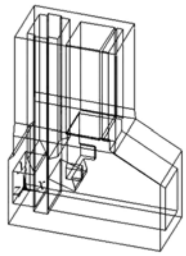

Fig. 4 Importing without defeaturing

3 Analysis of temperature field of intake tower

The temperature analysis of the intake tower during the construction period involves aspects of the temperature 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 (Gong and Xie 2004). This study carried out a simulation analysis of the temperature field considering nearly all conditions of construction, using the birth-death element and programming with APDL.

3.1 Solving temperature field principle

3.1.1 Unsteady temperature field analysis

The temperature of concrete changes during the construction period due to the effect of hydration heat of cement. This problem can be expressed as a heat conduction problem with internal heat sources in the area. The unsteady temperature field $T(x, y, z, \tau)$ is written as (Zhu 1999):

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

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

In the 3D unsteady temperature field analysis, the functional form $I^{\mathrm{e}}(T)$ is

$$I^{\mathrm{e}}(T)=\iiint_{\Delta R}\left\{\frac{1}{2}\alpha\left[\left(\frac{\partial T}{\partial x}\right)^2+\left(\frac{\partial T}{\partial y}\right)^2+\left(\frac{\partial T}{\partial z}\right)^2\right]+\left(\frac{\partial T}{\partial \tau}-\frac{\partial \theta}{\partial \tau}\right)T\right\}\mathrm{d}x\mathrm{d}y\mathrm{d}z$$
$$+\iint_{\Delta D}\bar{\beta}\left(\frac{1}{2}T^2-T_{\mathrm{a}}T\right)\mathrm{d}s \tag{2}$$

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

3.1.2 Initial conditions and boundary conditions of concrete

The initial conditions are the distribution laws of the initial transient temperature of internal concrete. The calculated initial temperature of concrete is 10 ℃.

The index formula of hydration heat of cement is

$$Q(t)=71\ 610\left[1-\exp(-0.36t)\right] \tag{3}$$

where t is the pouring time. The conversion between Q and θ is

$$\frac{\partial\theta}{\partial\tau}=\frac{Q}{c\rho} \tag{4}$$

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

$$-\lambda\left(\frac{\partial T}{\partial n}\right)=\beta\left(T-T_{\mathrm{a}}\right) \tag{5}$$

where n is the normal direction. Both T_{a} and β are constants or variables (Ashida and Tauchert 1998; Lin and Cheng 1997).

During the maintenance period, the insulation materials of concrete are steel formworks and straws, and the exothermic coefficient of the outer surface is reduced as equivalent processing. The exothermic coefficients of the steel formwork and the straw are 45 kJ/(m^2·h·℃) and 10 kJ/(m^2·h·℃), respectively.

Based on the local temperature during construction, the following formula can be fitted according to the temperature variation curve:

$$T=26.1-25.1\cos\left[\frac{\pi}{284}(t-79)\right] \tag{6}$$

3.2 Analysis of temperature field in ANSYS

The simulation scheme of layered construction, which is based on the real construction scheme, is shown in Table 2. The pouring days in Table 2 are all the total days of construction for each layer. A layer is not poured until the former layer is poured.

Table 2 Simulation 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

The feature points are selected in every layer above the base plate. The maximum temperatures and the temperature curves are given in Table 3 and Fig. 5, respectively.

Table 3 Coordinates and maximum temperature of feature points

Feature point number	*x*	*y*	*z*	Maximum temperature (℃)
2	7.4	6.0	1.0	24.268
3	8.4	9.9	5.0	24.353
4	16.4	16.0	5.0	25.743
5	8.4	20.0	5.0	24.346
6	16.4	25.0	8.0	26.611
7	1.0	30.0	2.0	29.224
8	8.4	34.5	5.0	29.934

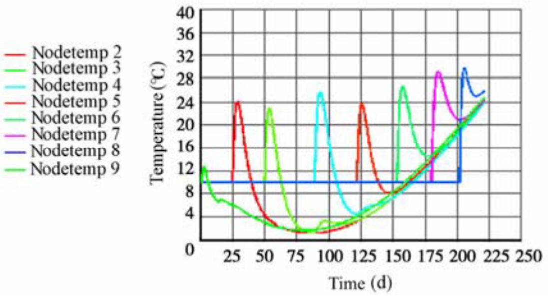

Fig. 5 Maximum temperature curves

Fig. 5 shows that the maximum temperature of each layer occurs on the 3rd or 4th day after pouring, and then the temperature decreases with time, which is consistent with related literature (Lin and Cheng 1997; Luna and Wu 2000; Wu and Luna 2001). In Fig. 5, the numbers of feature points from 2 to 8 are corresponding to their maximum temperature curves from Nodetemp 2 to Nodetemp 8, and the curve of Nodetemp 9 is the air temperature curve. Feature point 8, the maximum temperature of which is 29.934 ℃, occurring on the 206th day of the total construction period, shows the maximum temperature rise during the construction period. Feature point 4, the coordinates of which are (16.4, 16.0, 5.0), shows the maximum temperature difference of 23.534 ℃.

4 Analysis of thermal stress of intake tower

Expansion or contraction of the structure occurs during heating and cooling. If the expansion or contraction of different parts is inconsistent, then thermal stress occurs. The indirect method was adopted in this study: the temperature of nodes was first obtained in analysis of the temperature field, and then applied to the structure as a body load.

4.1 Selection of calculating parameters

The parameters of concrete are given in Table 4.

The elasticity modulus is

$$E_1 = 3.6\times10^{10}\left[1-\exp\left(-0.40t^{0.34}\right)\right] \tag{7}$$

Table 4 Parameters of concrete

Material	Density (kg/m^3)	Coefficient of linear expansion (1/℃)	Poisson ratio
Concrete	2 447.5	9×10^{-6}	0.167

The creep effect must be considered in analysis of temperature stress. The creep degree of concrete is influenced by the cement type, water-cement ratio and admixture. The formula of the creep degree is

$$C=\left[0.23\left(1+9.2t^{-0.45}\right)\left(1-e^{t_1}\right)+0.52\left(1+1.17t^{-0.45}\right)\left(1-e^{t_2}\right)\right]\times10^{-10}/3.60 \tag{8}$$

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

$$E=E_1/\left(1+CE_1\right) \tag{9}$$

4.2 Analysis of thermal stress in ANSYS

As in analysis of the temperature field, feature points were selected in each layer above the base plate, and their coordinates were the same as those in the temperature field analysis. The maximum thermal stress of each point is shown in Table 5. Feature point 9, the coordinates of which are (17.4, 10.8, 8.0), is the point with the maximum thermal stress.

Table 5 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 Fig. 6.

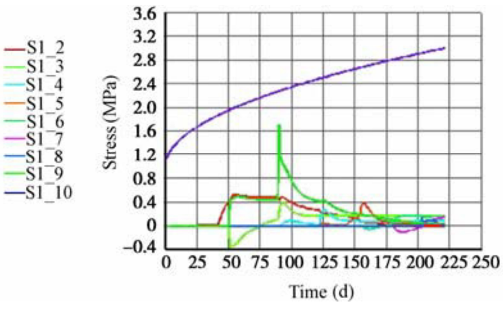

Fig. 6 Maximum stress curves

In Fig. 6, the numbers of feature points from 2 to 9 are corresponding to their maximum stress curves from S1_2 to S1_9, and the S1_10 curve is the ultimate tensile stress of concrete. The formula of concrete's ultimate tensile stress is

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

The figures and table show that the maximum thermal stress of the intake tower is 1.68 MPa, occurring on the 90th day of the construction period, which is the end of the third layer maintenance period and the beginning of the pouring of the fourth layer. It is known that the thermal stress increases with the temperature difference.

Feature point 9 is located at the interface between the third layer and the fourth layer. Thus, it is postulated that the maximum thermal stress is caused by the instantaneous temperature difference between two layers in the pouring period. In Fig. 6, the S1_10 curve shows the ultimate tensile stress curve of concrete. It is known that the maximum thermal stress of each point in the intake tower during the construction period is less than the ultimate tensile stress of concrete.

5 Conclusions

(1) The problem of the interface between SolidWorks and ANSYS is resolved in this study, realizing an effective combination of the advantages of both SolidWorks and ANSYS and providing a basis for analysis in ANSYS.

(2) Using a birth-death element and considering layered construction, dynamic boundary conditions, hydration heat, the dynamic elasticity modulus, autogenous volume deformation and creep of concrete, the temperature field and thermal stress during the construction period are conveniently obtained due to the virtues of secondary development with APDL.

(3) The analysis of temperature shows that the temperature of concrete rises rapidly in the early stage of construction, reaches a maximum value of 29.934 ℃ on the 3rd or 4th day after pouring, drops thereafter, and is consistent with air temperature after about 30 days. The thermal stress increases with the temperature difference, and the occurrence time of the maximum thermal stress is consistent with that of the maximum temperature difference. The maximum thermal stress occurs at the interface of new and old layers and is caused by the instantaneous temperature difference, the value of which is 1.68 MPa.

(4) The maximum thermal stress is less than the ultimate tensile stress of concrete, which illustrates that the curing measures in construction are effective. Meanwhile, in view of the fact that the maximum thermal stress occurs at the interface of new and old layers, more attention should be paid to it, especially when there is a long interval of time between the pouring of different layers.

References

Ashida, F., and Tauchert, T. R. 1998. An inverse problem for determination of transient surface temperature from piezoelectric sensor measurement. *Journal of Applied Mechanics*, 65(2), 367-373. [doi:10.1115/1.2789064]

Barrett, P. R., Foadian, H., James, R. J., and Rashid, Y. R. 1992. Thermal-structural analysis methods for RCC dams. *Proceedings of the Conference of Roller Concrete III*, 407-422. San Diego: ASCE.

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

Kawaguchi, T., and Nakane, S. 1996. Investigations on determining thermal stress in massive concrete structures. *ACI Materials Journal*, 93(1), 96-101.

Lin, J. Y., and Cheng, T. F. 1997. Numerical estimation of thermal conductivity from boundary temperature measurements. *Numerical Heat Transfer*, 32(2), 187-203. [doi:10.1080/10407789708913887]

Liu, L. J., and Ren, J. P. 2005. Application of the secondary development in SolidWorks. *Mechanical Management and Development*, (1), 74-75. (in Chinese)

Luna, R., and Wu, Y. 2000. Simulation of temperature and stress fields during RCC dam construction. *Journal of Construction Engineering and Management*, ASCE, 126(5), 381-388. [doi: 10.1061/(ASCE) 0733-9364(2000)126:5(381)]

Tatro, S. B. and Schrader, E. K. 1985. Thermal consideration for roller compacted concrete. *ACI Structural Journal*, 82 (2), 119-128.

Wu, Y., and Luna, R. 2001. Numerical implementation of temperature and creep in mass concrete. *Finite Elements in Analysis and Design*, 37(2), 97-106. [doi:10.1016/S0168-874X(00)00022-6]

Zhang, J. 2005. Interface design between AutoCAD and ANSYS. *Chinese Quarterly of Mechanics*, 26(2), 257-262. (in Chinese)

Zhu, B. F. 1999. *Thermal Stresses and Temperature Control of Mass Concrete*. Beijing: China Electric Power Press. (in Chinese)

Zhu, B. F., and Xu, P. 2001. Methods for stress analysis simulating the construction process of high concrete dams. *Dam Engineering*, 6(4), 243-260.

Zhu, B. F. 2006. Current situation and prospect of temperature control and cracking prevention technology for concrete dam. *Journal of Hydraulic Engineering*, 37(12), 1424-1432. (in Chinese)