Redevelopment of ANSYS

in Mechanics Analysis of Railway-highway Combined Bridge

Yiping Yin

College of Civil Engineering and Mechanics

Hua Zhong University of Science and Technology

Wuhan 430074, China

Tel: 86-027-8755-2231   E-mail: yinjw@yahoo.cn

Weishi Liu

Chaling Construction Bureau

Hunan 412400, China

Abstract

The length of railway-highway combined bridge is large and it's calculating model having many elements makes it need a great of manpower and time in the Finite Element Analysis (FEA). With redevelopment of ANSYS, a FEA program is empoldered so that complex load calculation and loads combination is worked out automatically in ANSYS. It reduces manpower, shortens time and improves work efficiency. the FEA program was used successfully in analysis of Zhengzhou Yellow River railway-highway combined bridge, and all the work of calculation and results processing can be finished in sixteen days, which saves much time in researching and designing bridge and obtains good social benefits and economic benefits. It also provides convenience for other analysis of railway-highway combined bridges and demonstrates idea for other similar questions of development of ANSYS.

Keywords: ANSYS, Redevelopment, Railway-highway combined bridge, FEA

1. Introduction

The integral spatial mechanics analysis of bridge structure needs a variety of load calculation, including the dead load, vehicles and crowd load, bearing sedimentation load, site wind load and extreme wind load, temperature load and seismic load, and so on. The analysis also should consider the loads combination, including the dead loads combination, the main loads combination and the additional loads combination, and identify the most disadvantaged loads combination. Given importance and complexity of the bridge analysis, the Finite Element Method is often used instead of the manual calculation.

Some bridge analysis software, such as Bridge Doctor, 3D3S, 3D-BRIDGE, are widely used in China now, but the MIDAS, a Korea bridge analysis software, has the largest number of users. These software are major to bridge with great convenience, it is limited only to the general scale of the bridge structure analysis. ANSYS (SAS IP,1998) is a good general-purpose finite element analysis software can solve large-scale problems. However, it is difficult to solve the professional structure problem of bridge directly, which costs much manpower to carry out loads combination and results processing. So sometimes the redevelopment of ANSYS for concrete problem of bridge structure is needed (CHEN Jin,2003; YE Meixin,2005).

In 2005 the authors of this article used ANSYS to calculate integral spatial stress of cable-stayed bridge in Ningbo City Airport Road, which has degrees of freedom less than 80,000. It took more than 50 days to finish the calculation and get the detailed researching report.

That the large length of railway-highway bridge with lots of components makes the researchers spend a long time in doing the FEA work, however, the researching and designing time is now very urgent, so the FEA can not be worked by using conventional bridge professional software or artificial results collation of the analysis of ANSYS software.

The investigation commissioned by the China Zhongtie Major Bridge Reconnaissance & Design Institute Co., Ltd., we developed a FEA program by redevelopment of ANSYS to calculate the integral spatial mechanics of railway-highway

combined bridge, making tedious analysis of the structure, combination of loads and collation of results in ANSYS completed automatically. The use of the program in the FEA of Zhengzhou Yellow River railway-highway combined bridge (China Zhongtie Major Bridge Reconnaissance & Design Institute Co., Ltd.,2007) (about 14,600,000 degrees of freedom) costs only 16 days to finish the calculation and get a satisfactory report. Development of the program has greatly shortened the researching and designing time of the bridge, rapidly improved the efficiency, provided a convenient analysis to other railway-highway combined bridge, and took a example for similar problems in redevelopment of ANSYS.

2. Redevelopment of ANSYS

Developed by ANSYS company, ANSYS software was the first design analysis software that passed through the ISO9001 quality certification. The finite element analysis software uses advanced technology, and at the same time it has a good pre-processing and post-processing functions. ANSYS is a powerful and practical tool for the calculation of very complex regulatory structure.

2.1 User Interface Design Language

ANSYS software provides a dedicated language, UIDL (User Interface Design Language), for user interface design. UIDL is a procedural language, which allows users to change ANSYS graphical user interface (GUI) and some of the groups. UIDL provides a powerful tool, which allows users to use flexibly, to organize GUI according to personal preferences. It plays a role that can not be underestimated in reorganizing commands, erecting bridges between ANSYS and other software.

2.2 ANSYS Parametric Design Language

APDL stands for ANSYS Parametric Design Language, a scripting language that you can use to automate common tasks or even build your model in terms of parameters (variables). APDL also encompasses a wide range of other features such as repeating a command, macros, if-then-else branching, do-loops, and scalar, vector and matrix operations, particularly worth mentioning of which is macro. You can record a frequently used sequence of ANSYS commands in a macro file (these are sometimes called command files). Creating a macro enables he to, in effect, create your own custom ANSYS command. In addition to executing a series of ANSYS commands, a macro can call GUI functions or pass values into arguments.

While APDL is the foundation for sophisticated features such as design optimization and adaptive meshing, it also offers many conveniences that you can use in your day-to-day analyses. As you become more adept at the language, you will see that the applications for APDL are limited only by your imagination.

2.3 User-Programmable Features

User-Programmable Features (UPFs) are ANSYS capabilities for which you can write your own FORTRAN routines. UPFs allow you to customize the ANSYS program to your needs, which may be a user-defined material-behavior option, element, failure criterion (for composites), and so on. You can even write your own design-optimization algorithm that calls the entire ANSYS program as a subroutine. In fact, some standard ANSYS features began as user-programmed features. Typically, you can obtain good results with the ANSYS program when you exercise documented features using standard, recommended procedures. In some cases, however, you may need to employ nonstandard procedures.

3. Redevelopment of ANSYS in railway-highway combined bridge

ANSYS used in mechanics analysis of railway-highway combined bridges has some difficulties: load calculation, loads combination and results collation. Here we will use the redevelopment technology of ANSYS to address these difficulties one by one.

3.1 Load calculation

3.1.1 Dead load

Calculating the material density according to the first period dead load, and defined the material density of the bridge deck based on second period dead load:

mp,dens,1,125350   ! material density of the railway bridge deck

mp,dens,5,3397   ! highway bridge deck

mp,dens,2,9800   ! other material

3.1.2 Vehicle load

Acting a moving point load and considering the reduction factors. The value of the moving point is calculated by the total vehicles load on railway bridge and highway bridge respectively:

```
*do,i,0,1488,1                          ! load moving by loop
fdele,all,all                           ! delete the original loads
nsel,s,loc,x,i*0.5                      ! select location for acting load
f,all,fy,-1.0*Fcor*1e4                  ! acting unit point load
*enddo                                  ! Fcor is the reduction factor
```

3.1.3 Bearing sedimentation load

The value of settlement at intermediate piers is 0.015m and 0.010m at side piers:

```
d,n1,uy,-0.015                          ! intermediate pier
d,n2,uy,-0.010                          ! side pier
```

3.1.4 Temperature load

Increasing and decreasing temperature in bridge is 20℃, in components is 10℃:

```
    Allsel                              ! select all elements in bridge
bfe,all,temp,,-20                       ! decreasing temperature
esel,s,elem,, 1,140000                  ! select elements in highway bridge deck
bfe,all,temp,,-10
```

3.1.5 Site wind load

Includes positive longitudinal wind and negative longitudinal wind, left transverse wind and right transverse wind:

```
f,n1,fz,-1180                           ! right transverse wind
f,n2,fx,-1180                           ! negative longitudinal wind
```

3.1.6 Vehicle braking load

Calculated based on you lane load acted on railway bridge and highway bridge respectively and considering the coefficient to brake:

```
nsel,s,loc,z,4.5
nsel,r,loc,y,0
f,all,fx,-4.8e3                         ! braking load in railway bridge
nsel,s,loc,z,4.5
nsel,r,loc,y,14
f,all,fx,-2.1e3                         ! braking load in highway bridge
```

3.1.7 Extreme wind load

Direction can be referred from the part 5): site wind load. This item must not combine with vehicle load:

```
f,n1,fz, -3072                          ! right transverse wind
f,n2,fx, -3072                          ! negative longitudinal wind
```

3.2 Loads combination

There are eight groups of loads combination:

1) Dead load combination:dead load

2) Main loads combination:dead load+ vehicles and crowd load + bearing sedimentation load

3) Additional loads combination 1:main loads combination +increasing temperature combination
+ transverse wind load + vehicle braking load

4) Additional loads combination 2:main loads combination +decreasing temperature combination
+ transverse wind load + vehicle braking load

5) Additional loads combination 3:main loads combination +increasing temperature combination
+ longitudinal wind load + vehicle braking load

6) Additional loads combination 4:main loads combination +decreasing temperature combination
+ longitudinal wind load + vehicle braking load

7) Additional loads combination 5: dead load+ bearing sedimentation load + extreme wind load

8) Seismic loads combination: dead load+ bearing sedimentation load + seismic load

Procedures for vehicle loads combination in highway are as follows:

```
*do,i,0,1488,1
    Lcdef, i+1, i                              ! time i represents load (i+1)
*enddo
max=0                                          ! compare the results to get the
min=0                                          ! maximum value and minimum value
*do,i,0,1488,1
        *if, Lcdef(i+1),gt,0,then
            max =max + Lcdef(i+1)*1.6785        ! vehicle load coefficient is 1.6785
        *else
            min =min + Lcdef(i+1)*1.6785
        *endif
    *enddo
```

3.3 Results collation

The results include displacement (ux, uy, uz), reaction force, member forces (axial force, in-plane shear, in-plane bending moment, out-plane shear, out-plane bending moment), stress of bridge decks (normal stress and shear stress), cable force and cable stress amplitude. As the number of the bridge components is very large, it is very cumbersome to collate the results by manpower.

Here APDL is used to collate the results. Attaching statement heading to the calculating data can get reporting results, then output the reporting results to EXCEL software to form researching report.

Procedures to get calculating data of reaction force:

```
*dim,Reaction_fy, , 6                          ! dimension Reaction_fy
*do,i,1,6,1
*get, Reaction_fy(i), node, I, rf, fy          ! save reaction force to dimension
*enddo
    Reporting data of member forces:
*do,i,1,3960,1                                 ! F_beam_C(i) is calculating result
   F_beam_R(k)= F_beam_C(i)                    ! F_beam_R(i) is reporting result
*if,mod(i,5),eq,0,then                          ! Each table consists of 5 rows data
k=k+6                                          ! 5 rows interval between two tables:
*else                                          ! 2 rows space and 3 rows heading
   k=k+1
*endif
*enddo
```

4. Calculation example

The proposed Zhengzhou Yellow River railway-highway combined bridge will become the world's longest railway-highway combined bridge. The main bridge is designed with six consecutive cable-stayed steel truss bridges with 168m. In FEA, the mid three span can be combined into only you span for easy calculation, so the calculating model has five spans, however the bridge is seven spans in fact. That manual calculating load, combining loads and collating results in usual practice not only encounters inevitable human errors, but also it will costs too much time to get a detailed researching report. Here the redevelopment of ANSYS is used to the analysis, so that complex load calculation and loads combination is worked out automatically in ANSYS. It reduces manpower, shortens time, improves work efficiency and avoids man-made errors.

The FEA models of Zhengzhou Yellow River railway-highway combined bridge are in figure 1, and some calculating results are in Figure 2.

5. Conclusions

With redevelopment of ANSYS, a FEA program is empoldered so that complex calculation of internal force analysis and cases combination is worked out automatically in ANSYS. It reduces manpower, shortens time, improves work efficiency and avoids man-made errors, as well as demonstrates idea for other similar questions of development of ANSYS.

References

ANSYS APDL Programmer' s Guide[S]. USA: SAS IP,1998.

CHEN Jin, XIAO Rucheng, JIANG Jianjing. Probabilistic determination for initial cable forces of cable-stayed bridges under dead loads [J]. China Civil Engineering Journal, 2003,36 (12):58 - 63.

The Third Survey and Design Institute of China Railway, China Zhongtie Major Bridge Reconnaissance & Design Institute Co., Ltd.. Design files for Zhengzhou Yellow River railway-highway combined bridge [R], 2007.

YE Meixin, HAN Yanqun, ZHANG Min. Development and application of ANSYS in long - span cable - stayed bridge[J]. Journal of railway science and engineering, 2005, 2(5):56-59.
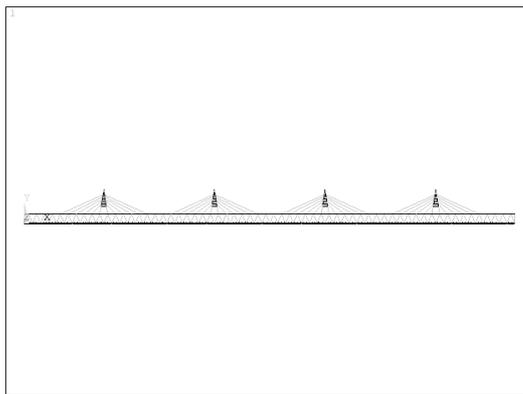

Figure 1.a   bridge model (front view)
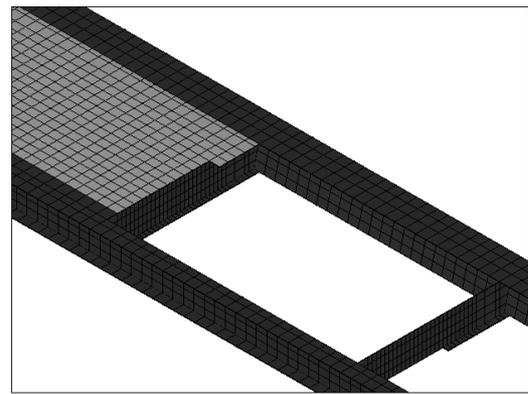

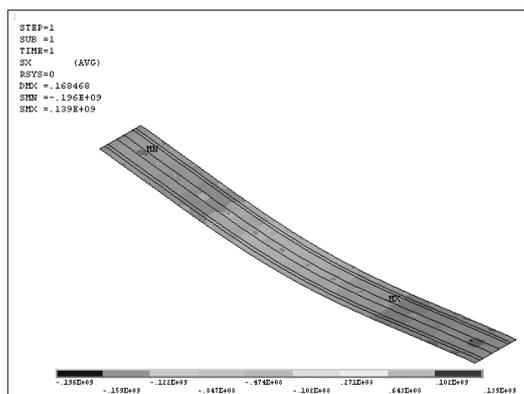Figure 1.b   elements of railway bridge deck
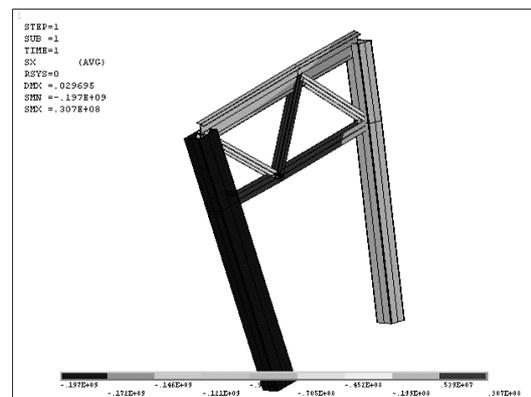

Figure 2.a   stress of railway bridge deck(:Pa)

Figure 2.b   stress of frame cushion of bridge(:Pa)