Dynamic Simulation of A Large Deployable Space Structure with Pinned Joints

Wen Rong

Beijing Institute of Astronautical System Engineering
P.O.Box 9208 , Beijing 100076 , P. R. China
Email: wrongrc@public.bta.net.cn

Abstract:

Plays of joints strongly influence the dynamics of the large deployable space structures. In this paper, dynamic simulation of space structures with pinned joints is discussed. The study can be simply divided into two parts: the flexible effect of structure is simulated by MSC/NASTRAN, and the global motion of rigid body by DADS. The details of how to select the deformation modes in MSC/NASTRAN and how to deal with the digital problems in DADS will be given.

Keywords: Space Structures Gap Deployable Nonlinear MSC/NASTRAN DADS

1. Introduction

Most deployable space structures are constructed from truss and plate elements with pin joints. Due to the manufacturing tolerance, small play or looseness inevitably exits in the joints. Even though an individual pin connection has very small gap, the whole structure with lots of joints has accumulated looseness. Therefore, the practical structure differs from an ideal one, Its position is inadequate within the scope of theoretical value. To some extent, accumulated plays increase the number of DOF(Degree of Freedom) and strongly influence the dynamics of the structure. When the normal load in the joints is big or external forcing is small, the dynamical nonlinearity would be obvious even if the structure has only a microslip.

Dynamical behavior of nonlinear system is much more complicated than that of linear's. Main methods used in studying the nonlinear systems are qualitative method and quantitative method. The former could give the individual features of the system in a phase space. It is useful to know the global characteristics, but the accurate solutions can be gained only by the quantitative methods. Because of the complication of nonlinear system, two methods are often combined to solve the problem.

Dynamics of deployable space structures with pinned joints can be divided into two parts. One is flexible multibody dynamics, another is the problem of gaps between joints.

The popular model of play between joints is Dubowsky Model or Modified Dubowsky Model, however, this can easily causes some digital problems. Therefore, the pin-jointed structures should be modeled carefully with DADS. The main purpose of MSC/NASTRAN is to calculate the "Deformation Mode" or "Deformation space" of structures. In another words, MSC/NASTRAN should give all of probable deformation types, and the practical deformation is linear combination of these deformation types.

2. Flexible Multibody Dynamics

2.1 Dynamic Equations

Space deployable structure is a flexible multibody system, which cannot to be generally resolved with analytical resolution. Approximate method must be adopted to gain numeric value.

Dynamic equations of multibody system are like this:

$$\begin{cases} M\ddot{q} + f_q^T\, l = F \\ f(q,t) = 0 \end{cases} \tag{1}$$

In which, M stands for general mass matrix, q for general coordinates, $\dot{q}$ for general velocity, $\ddot{q}$ for general acceleration, F for general force vector, f for constraint equations, f_q for Jacobian matrix of constraint equations, l for Lagrange multiplier.

To resolve equations (1), we deviate constraint equations twice, so that

$$\begin{bmatrix} M & f_q^T \\ f_q & 0 \end{bmatrix} \begin{bmatrix} \ddot{q} \\ l \end{bmatrix} = \begin{Bmatrix} F \\ g \end{Bmatrix}, \quad g = -(f_q\dot{q})_q\dot{q} - 2f_{qt}\dot{q} - f_{tt} \tag{2}$$

It can be considered as algorithm equations of $\ddot{q}$ and l. After $\ddot{q}$ and l gained, $\dot{q}$ and q can be resolved by integration method. Therefore, the state variables (q, $\dot{q}$) of system are determined at every time t.

2.2 Description of Deformation

The most important and difficult problem of flexible multibody dynamics is the description of flexibility. As we know, the DOF (Degree of Freedom) of a flexible body is infinite, which must be reduced during numerical simulation. There are three methods which can describe the deformation : LPM(lumped parameter modal), AMM(assumed mode method) and FEM (finite element method).

LPM considers the system as rigid body system, and the flexibility is modeled as spring-damper. This method avoids the description of deformation of flexible body. However, equivalent principle must be carefully selected to get the stiffness of spring and the damping of damper. For some systems with low structure frequency and large displacement, LPM is a good choice.

For AMM, the deformation of flexible body is regarded as linear combination with some assumed modes. In other words, assumed modes construct the deformation space of flexible body, and the modes can also be called base function. This is similar with structural dynamics.

FEM is the extension of AMM. The modes of system with complicated geometry and boundary conditions can be gained easily by FE analysis. There are three basic modes, which are usually adopted: normal mode, static mode and constraint mode.

All modes are the probable deformation forms of flexible body. Normal mode is the result of FE modal analysis, which decides the inner characteristics of structure. Static mode and constraint mode are essentially the static analysis of FE model.

As we know, FE modal analysis is much enough to describe the deformation of a structure with certain boundary conditions. However, it is not true everywhere. For flexible multibody system, FE modal analysis cannot describe the local joint-induced deformation. It also needs static mode and constraint mode. In fact, even the three modes are not necessary. Speak shortly, numerical simulation is hardly converged only with the normal mode, and convergent rapidly with three modes together in most cases. In some special cases, such as large deformation etc., numerical resolution cannot be converged.

2.3 Solution of Dynamic Equations

Simulation of flexible multibody system always induces numerical problems. The key points are stiff and errors accumulation.

General coordinates q in equations (1) include coordinates of the rigid motion and elastic motion. Comparing to the rigid motion, elastic one often changes more rapidly. The coupling between these two motions makes eigenvalue of coefficient matrix of dynamic equations widely spread, and the maximum is far away from the minimum. Therefore, the time step of integration maybe very small, which makes simulation convergent slowly. This is called stiff problem.

Most of the modeling theory and numerical algorithm of dynamic system are based on the Lagrange system. Its algorithm is ‘dissipation’, which cannot satisfy the conservation law. That means the resolution inevitably makes calculation errors in theory. Increasing with the integrating times, the numerical errors accumulate more and more. This is called errors accumulation.

From above we can see, the numerical simulation of dynamic system is not easy at all. A precise and efficient integration method is considerably important to the simulation.

3. Gap Problem

3.1 Description

Gap analysis begins with the mechanical system. In engineering practice, every type of imperfect characteristics, such as clearance, backlash, link flexibility etc., strongly effect the dynamic behavior of system. The affection ordinarily shows as vibration and noise. Especially, when the system or the component moves at a high speed, the behavior of system often looks like a random one’s.

The early study of gap problem often makes strong assumption. One method neglects the gap of linkage, and only considers the flexibility and friction between joints. It is equivalent to link the component with spring-damper. Another method considers the gap between joints, however, the gap is regarded as rigid one. The advantage of these two methods is simple, and can be easily resolved. The shortage is that only the momentum, not force, can be gained in resolution process. In fact, the contact force between joints normally determines the complicated behavior of system.

3.2 Classic Method

Gaps always cause contact between joints. The physical process of contact is very complicated. To simplify the problem, classic method often makes such assumptions:

(1) Contact process finishes during an infinite time interval, and every component can be modeled as rigid body at contact time.
(2) Contact surface can be considered as a point, and the point does not change during the contact process.
(3) Dry friction does not exist.
(4) Defining restitution coefficient e as the ratio between impulse of restitute process and that of compress process. If e=1, the contact is perfect elastic, e=0, complete imperfect elastic. The restitution coefficient does not change during contact process.

Above assumptions is the base of classic method of contact problem. Based on assumption (1), time interval of contact process is infinite: $\Delta t \to 0$. Due to the velocity and angle velocity of all bodies are finite, the position and orientation keep unchanged during contact. So as to the spring and damper, the force and moment make no affection. Only the contact force and contact moment cause the increase of velocity and angle velocity of system. Define contact impulse as:

$$\hat{F} = \lim_{\Delta t \to 0} \int_{t_0}^{t_0+\Delta t} F(t)dt \qquad (3)$$

And also define the contact impulse moment as the moment of contact impulse. Because the constraint react force is dependent with the contact force, constraint react impulse exists in the joints. So, the contact problem can be resolved by resolution of two groups of variables: one is instantaneous increase of velocity of system, another is impulse and impulse momentum at contact point and constraint joints.

3.3 Analytical Method

For flexible multibody system, above assumption (1) is not always satisfied, especially when the flexible deformation is large. That means the contact force at contact point is a function of time. Logical function method is usually used to deal with the case. Assume the contact force is like this

$$\begin{aligned} F^I &= F(q,\dot{q}) * L(S(q)) \\ &\text{When } S(q) < 0 \text{ , } L = 1 \\ &\text{When } S(q) \geq 0 \text{ , } L = 0 \end{aligned} \qquad (4)$$

In which, L is a logical function, L=1, contact occurs, $F(q,\dot{q})$ is contact force function. For example, the function can be selected as this

$$F^I(q,\dot{q}) = KS(q) + C\dot{S}(q) \qquad (5)$$

In which, K is stiffness of spring, C damping coefficient. Of course, K must be big enough to avoid the penetration between contact bodies. Due to the analytical function of contact force, the system is an ordinary multibody dynamic system, and can be resolved by ordinary methods

Most methods of large flexible multibody system are similar with logical function method. The impact pair model, which built by S.Dubowsky and F.Freudenstein in 1971 to study the contact between pinned joints, is a famous example.

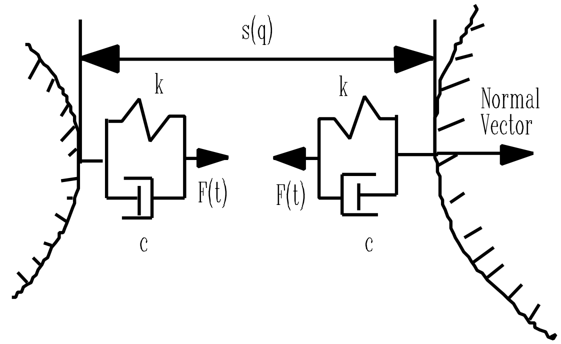

Fig.1 Logic Function Model

In order to see clearly, consider a single degree of freedom(1-DOF) system, in which the slip exits only along one direction. Assume sliding friction between joints is Coulomb friction:

$$f_N = mN \tag{6}$$

where m is friction factor, N the normal pressure

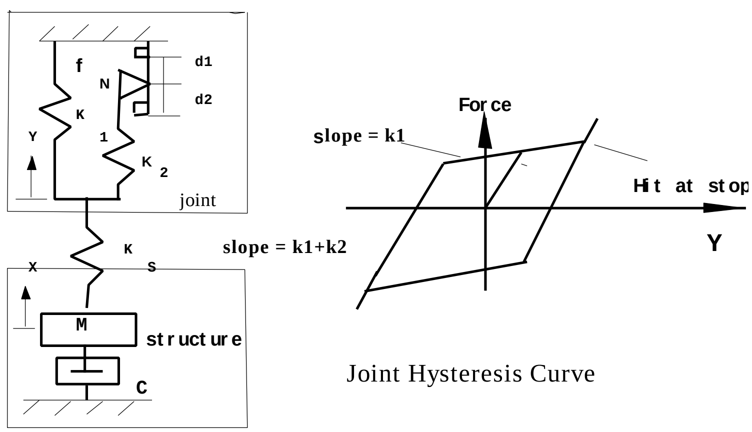

Fig.2 Impact Pair Model

When axial force is smaller than f_N, or $Y < mN / k_2$, there is no relative slide between joints, joint stiffness is $k_1 + k_2$. When axial force is bigger than f_N, or $Y > mN / k_2$, slide between joints occurs. If the gap $d = d_1 + d_2$ is long enough, joint is plastic, stiffness is k_1,if the gap is small, system would be elastic hardening at the end of gap, then stiffness is $k_1 + k_2$.

4. Simulation with MSC/NASTRAN and DADS

In this section, we will discuss how to resolve large deployable space structures with pinned joints. MSC/NASTRAN is used to get the basic modes of structure, and DADS to describe the gap and dynamic behavior.

As mentioned before, there are three types of modes that will be needed. The most important mode is normal mode, which gives the basic deformation space of structure. In fact, some software (such as ADAMS) only adopts normal mode. There is not sufficient reason to say it is right or not. However, a large amount of numerical tests show that simulation converges faster when three modes are taken together. From the view of appearance, the static and constraint modes do not harm to convergence. The static mode is the displacement result of nodes in reality.

It gives the actual deformation shape. Imaging a cantilever beam with a force centralized on the middle point, the deformation of beam is not easily combined by normal mode. For the static mode, it is easy. Because the static mode is just the displacement of the beam with the force acted on the middle. So does the constraint mode. Due to the relative motion between flexible bodies, the constraint is not fixed in space scope. It is necessary to consider the affection of constraint motion. Constraint mode is one of the static modes, which deformation is caused by the unit displacement along the constraint direction. In one word, the modes, which construct the deformation space of multibody system, are more like the actual deformation; the simulation converges more quickly.

Here gives a typical example of transformation from modal analysis of MSC/NASTRAN to DADS. The data file is like this:

File.dat

```
Assign output2='File.op2' unit=11
Assign output4='File.mgg' unit=12
ID Modal Beam
Sol 103
Time 1
$
$ Header For DADS
$
Compile IFPL  Souin=mscsou  Nolist  Noref
Alter 'GP1'
Output2 GPL,BDPDT,,,//-1/11 $
GP2    Geom2,EQEXIN,EPT/ECT $
Output2 ECT,,,,//0/11
$
Compile SEMG1 Souin=mscsou Nolist Noref
Alter 'GPWG'
Output2 OGPWG,,,,//0/11 $
Tabpt OGPWG//$
$
Compile Phase1a Souin=mscsou Nolist Noref
Alter 'End $ Phase1a'(1,-2)
Output4 MGG,,,,//-1/12 $
Output4 ,,,,//-2/12 $
$
Compile Moders Souin=mscsou Nolist Noref
Alter 'Endif $ Lanczos'
Output2 Lama,,,,//0/11 $
$
Compile Sedrcvr Souin=mscsou Nolist Noref
Alter 'Sdr2'(6,2)
Output2 Ougv1,,,,//0/11 $
Output2 ,,,,//-9/11 $
Cend
$
$ Case Control Segment
$
Displacement=all
Stress(Corner)=all
Method=13
Begin Bulk
Param,GRDPNT,0
Eigr,13,sinv,0.1,100.0,,2
…
EndData
```

According to our experience, the attach mode is very important to the convergence velocity of a system with revolute joints. Giving revolute degree of freedom a unit angle, the displacement can be

The gap contact is much more difficult to deal with than the flexibility. The problem is also focus on the convergence of simulation. Comparing to the dimension of rigid and flexible motion, the gap between joints is very small. So, the contact may occur frequently. When the contact force varies consistently with the time, the time step of integration should be same small. This is similar with the stiff problem of flexibility. One method is to look for a precise and efficient integration algorithm. Another method is to modify the contact model. As figure 3 shows, the contact force is not linear to the penetration distance yet. Exponential e should be bigger than 1, thus can make sure the contact force at contact point is c^1 continuous. If the force is linear, its differential should not be continuous at contact point. This often leads to the numerical divergence. Simulation shows that when the size of gap decreases, only modified contact model can converge.

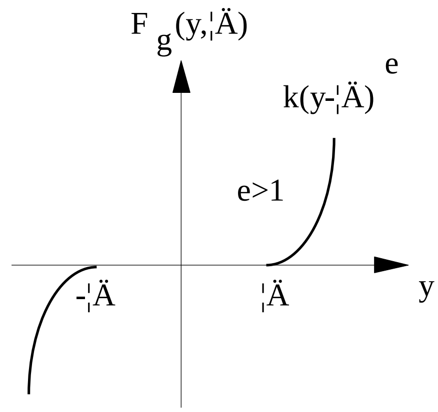

Fig. 3 Modified Contact Model

There are three types of contact force in DADS. The first is spring-damper force like the impact pair model. The second is Herz contact force. The last is user defined nonlinear contact force. The Herz contact model is recommended and default. The friction coefficient, restitution coefficient and Yang modulus of materials should be given. The most useful function in DADS/Contact is Point-to-Segment Contact. The segment can be any surface which is constructed in DADS.

The important parameters defined by the user are the Radius Point, Radius of Exclusion and Max Depth. The Max Depth is the maximum allowable distance that two contact bodies penetrate each other. The Radius Point is the radius of the Point Body. The explanation of the Radius of Exclusion in User's Guide of DADS is, "the radius of exclusion around the segment body triad origin. Contact will occur only if the contact point is within the radius of exclusion. If the distance from the segment body triad origin to the point body origin is less than the sum of the radius of the contact point plus the radius of exclusion. If the Radius of Exclusion is set to zero (default value), the radius of possible contact is infinite". The large Max Depth will probably dismiss the contact, and the small Max Depth perhaps leads to numerical divergence.

In practical engineering, the simulation of deployable space structures with pinned joints is more complicated than mentioned above is expensive. However, comparing to the expensive test, the cost of calculation is much cheaper and effective.

Reference

[1] R.Wen,D.L.Wu, Analysis of nonlinear dynamical characteristics of deployable space structures,47th International Astronautical Congress, October 7-11, 1996, Beijing, Paper Number IAF-96-A.7.02

[2] A.A.Shabana,Dynamics of Multibody Systems,John Wiley & Sons,New York,1989

[3] J.Wittenburg,Dynamics of System of Rigid Bodies,B.G.Teubner Stuttgart,1977

[4] S·T·Hsu? J·H·Griffin, How Gravity and Joint Scaling Affect Dynamic Response, AIAA Journal Vol. 27, No. 9, Sep. 1989

[5] F·C·Moon? G·X·Li, Experimental Study of Chaotic Vibrations in a Pin-Jointed Space Truss Structure, AIAA Journal Vol. 28, No. 5, May 1990

[6] Junjiro ONODA? Kunji MINESUGI, Semi-Active Vibration Suppression of Truss Structures by Coulomb Friction, AIAA-92-2270-CP

[7] Steven W·Shaw? Richard H·Rand, The Transition to Chaos in a Simple Mechanical

System, Int. J. Non-Linear Mechanics, Vol. 24, No. 1, pp. 41~56, 1989
[8] G·X·Li? R·H·Rand and F·C·Moon, Bifurcations and Chaos in a Forced Zero-Stiffness Impact Oscillator, Int. J. Non-Linear Mechanics, Vol.25,No. 4,1990
[9] David Bella and Michael Reymond,DMAP and Database Application in MSC/NASTRAN Seminar Notes(Version 68), MSC
[10] DADS User's Guide/Contact Modeling Guide(Version 9.0), CADSI