Parametric Modeling and Finite Element Static Analysis of Large Reflector Antenna Based on APDL and MATLAB

Qin Li
Institute of Intelligent Manufacturing, Panzhihua University, Panzhihua, Sichuan, 617000, China
Corresponding author’s e-mail: liqin103@126.com

Abstract. In this paper, a Large Reflector Antenna is taken as the research object and the Parametric Modeling of Antenna Structure is achieved by using APDL language in ANSYS, and the APDL command flow generation was completed with the help of MATLAB, which achieved the program of modeling and improved its efficiency. Meanwhile, the paper applies parameterization to restrain and load the static load of Antenna, and makes an analysis on the Finite Element Static. This provides a basis for the analysis of the stiffness, deformation and structure optimization of Antenna.

1. Introduction
With the continuous development of communication, broadcasting, radar, guidance, radio astronomy and other fields, antenna, one of the key equipment, is facing the tough demands. On one hand, antenna’s structure is more complex and the aperture size is bigger and bigger[1-2]. On the other hand, the performance indicators of antenna (such as Antenna gain, the Antenna Sidelobe and Antenna cross-polarization and so on) will be affected[3-4] due to the tough working environment, such as self-weight, wind load and sunshine. If parameterization method is used in modeling, the design efficiency of complex models can be improved and it is convenient to modify model. After the model is built, loading simulation, can verify the model’s correctness and predict the endurance capability of Antenna.

APDL—a FORTRAN-like programming language, has the function of general program language. Using the program language and macro technology of APDL to organize and manage ANSYS Finite Element Analysis command can achieve Parametric Modeling, loading, solution and parametric post-processing, which contributes to realize Parametric Finite Element Analysis overall process[5]. In the process of Parametric Analysis, the parameters can be simply modified so it is possible to repeatedly analyze various sizes and loads of various designing schemes or sequence product, which greatly improve design and analysis efficiency.

According to the Parametric Modeling and static simulation of a Large Reflector Antenna in ANSYS, the paper reached the conclusion about the cases of deformation of structure. It provided a basis for further optimization design.

2. Parametric Modeling of Large Reflector Antenna Based on APDL and MATLAB

2.1. Finite Element Modeling steps
The Finite Element Analysis steps of Antenna structure is shown in Figure 1, there are three phases:
2.1. The Finite Element Analysis steps of Antenna structure

(1) Modeling phase. According to the theory of mechanics, remove unnecessary details from the actual Antenna structure and abstract a mechanical modeling that conforms to the mechanical requirements and close to the actual ones; then, select the unit, process the boundary constraint, simplify the load, and finally obtain the Finite Element Model.

(2) Solving phase. Disperse the Finite Element Model to generate date file, and then solve the problem with the corresponding structural analysis program.

(3) Explaining the analysis phase. The solution obtained form (2), is just a collection of data. Only if combining these quantitative results with actual problems, the mechanical behavior of the actual structure could be obtained and with the feedback resulted from it to testify the rationality of the model and guide the structural design.

2.2. Antenna Parametric Modeling

There are 3 steps to establish the Antenna Model in ANSYS. Firstly, decompose the whole Antenna into several large parts (back frame, center body, the vice reflector, bracket, the main reflector); then establish the model in a certain order; finally generate a whole Antenna by connecting each part. The sequence of modeling is: back frame-center body-the vice reflector and bracket structure-main reflector, and each part contains many details, for example the back frame part contains: radial beam, ring beam, diagonal; center body contains: central circular ring and central cylinder; the main reflector part except for the reflective panel, also short beams that connect the panels to back frame. In the process of modeling of each part, it is necessary to set the unit type, the elasticity modulus, density and Poisson’s ratio of the materials; meanwhile, set the node number with the rule. The following part focuses on the modeling of the vice reflector and reinforcing bars to illustrate the process of Parametric Modeling.

This paper takes the 12-meter cassergain Antenna (the most commonly used double reflector antenna) as an example, its main reflector is a rotating paraboloid, and the vice reflector is a rotating hyperboloid. Sheet moulding structure is adopted to make the vice reflector panel. The back of the formed thin shell with reinforcing bars is an entity reflector. It consists of eight same size hyperbolic panels as shown in Figure 2.

The steps to establish the model of the vice reflector are:

The 1st step, select the appropriate point as the key point to fit hyperbola (the key point starts from number 400, the line starts from number 500), generating two hyperbolas with intervals of 45°, then a public intersection point will be formed in the center of the side, and connect the ends of the two curves to form a hyperbola as shown in Figure 2.

Generated by adopting APDL parameterize language, the main languages of the single side panel are:

numstr,kp,400
numstr,line,500
csys,0
k, 400, 0.000000000, 0.0000000, 3589.4000000
Step 2, conduct grid division on the surface generated in step 1, and copy in rotating to form a complete vice reflector, as shown in Figure 3.

Step 3, add reinforcing bars on the side reflector to improve its stiffness and ensure the precision of the shape surface. BEAM4 Beam Element is selected to simulate the force endured by the vice reflector.

There are two ways to generate reinforcing bars: independent generation and generation through the node formed by grid partition of the panel. The former is hard to guarantee the relations between reinforcing bars and the panel so it can not meet the demands. The latter is adopted in this paper. The circular ring structure of the reinforcing bars will generate a wave structure so the nodes on the edge of panel will be chosen to avoid this case. Compared to selecting the node on the panel by mouse, node numbers should be recommended. In order to make better use of the numbering rule of nodes, it is necessary to change the grid division of panel and the way to generate single-sided panel. The formation of a single vice panel composed of two hyperboloids A and B is shown in Figure 4. And the way is: Select the appropriate points on the original model and connect the two points into a circular arc, then cut off the original two hyperbolas by subtraction in Boolean operation to make into four lines segments, and then reconnect the lines that were deleted.

APDL statement is:

```
flst,2,2,4,orde,2
fitem,2,500
fitem,2,-501
lsbl,p51x,503
```

The order of grid division should be given special attention after regenerating the surface: divide hyperboloid B before A, which is beneficial to number the nodes. Copying the surface also needs to follow the order of grid division. The side panel after dividing is shown in Figure 5. Form Figure 3 and 5, it can be seen that in the case of the same size of the side reflector, the grid is denser and there are more nodes after changing the way of grid division and the outer circle provides more options for reinforcing point.

The rules to number the nodes should be found in establishing the reinforcing rib model, then programme in MATLA to generate command flow.

The rule of numbering node on the hyperbolic panel A of Figure 4 is shown in Figure 6.
In Figure 6, $a$ is the node starts number of a single panel, $b$ is the required number of grids division (the number of three sides are equal). The around areas should be numbered first, and then the secant lines of three areas, last the three internal areas (one to three). The thick arrow represents serial number of a single column while the thin arrow presents the direction of column increasing.

The rule of numbering node on the hyperbolic panel B of Figure 4 is shown in Figure 7.

With the rule of node numbering, the required command flow can be generated by MATLAB program. The main algorithm of MATLAB programme is as follows:

```matlab
A=[a+3/2*b+2+(n-1),a+4*b+(n-1)+b/2-2+(b/2-1)^2];
disp(['e, ',num2str(A(1)),', e, ',num2str(A(2))]);
for i=1:1:b/2-2
    A=[a+4*b+(n-1)+i/2*b-(i+1)+(b/2-1)^2,a+4*b+(n-1)+(i+1)/2*b-(i+2)+(b/2-1)^2;
    disp(['e, ',num2str(A(1)),', e, ',num2str(A(2))]);
end
A=[a+4*b+(n-1)+(b/2-1)/2*b-(b/2-1)^2,a+3*b+b/2-n];
disp(['e, ',num2str(A(1)),', e, ',num2str(A(2))]);
```

First, entering the node start number $a$, the required numbers of grid division $b$ of each side and the start position of reinforcing rib $n$, one segment (one-eighth-segment) of the command flow of reinforcing rib can be generated. Second, copy the results of MATLAB to the file of command flow. Third, generate the remaining stiffeners with EGEN command. Now the model of vice reflector panel and reinforcing rib are established.

The same modeling method is adopted to establish models of other parts. After connecting the related models, the Finite Element Model of Antenna is established. It is shown in Figure 8.
3. Finite Element Static Analysis
The application of static load to the antenna structure model and the simulation calculation can predict whether the model structure meets the requirement of stiffness and deformation, which provides the basis for structural optimization design.

The main load on the Antenna is: self-weight, wind force, temperature stress, the inertial load of an Antenna in motion, feed support load, etc. What kind of load the Antenna bears in the working process is related to the working environment. In this paper, mainly calculated two kinds of loads that self-weight and wind force, obtained the magnitude of the deformation. The main loads borne by the Antenna are: self-weight, wind force, temperature stress, the inertial load of an Antenna in motion, feed support load, etc. The loads Antenna borne are related to the working environment. This paper focuses on self-weight and wind force to calculate and obtains the magnitude of the deformation.

3.1. Self-Weight
The Antenna’s self-weight exists at any time so it should be taken into consideration. It includes the weight of reflector and the weight of various devices mounted on the reflector structure. The self-weight of unit area of reflector is related to its materials and structural forms and antenna’s size and the minimum working wavelength. In general, the shorter the working wavelength is, the better the stiffness of the structure are and the heavier the weight of per unit area is. When the working wavelength is fixed, the bigger the size of antenna is, the heavier the self-weight of antenna is. So self-weight is one of the main loads in stiffness design for a large scale of antennas.

Add a full constraint to the node below the inner ring of all radiant beams in APDL parameterized way (7 to 112, median interval 7, total 16 nodes):

```
nsel,s,node,,7,112,7,
d,all,all
```

Calculate the node displacement of the antenna in the two cases of elevation and pointing flat in ANSYS with consideration of self-weight load.

3.1.1. Elevation analysis. When the antenna is in elevation, the gravity direction displacement cloud map is shown in the Figure 9.

![Figure 9. Elevation Analysis of displacement map](image)

From the results of calculation, when the antenna is in elevation, the maximum displacement 2.686mm, appearing on the outermost beam of the radiant beam, the direction is gravity direction, the whole deformation presented a central symmetrical distribution, conformed the actual deformation situation.

3.1.2. Refers to the flat analysis. When the antenna is in pointing flat, the gravity direction displacement cloud map is shown in the Figure 10.

![Figure 10. Analysis of displacement map flat](image)

According to results of calculation, when the antenna is in pointing flat, the maximum displacement in the direction of gravity is 5.374mm with the consideration of self-weight, and it appeared on the side of the vice reflector. the main reason is that the integral stiffness of the vice reflector bracket is not enough, the force of deformation on the bracket is added to the deformation of
the vice reflector itself. The problem can be solved by changing the cross section or structural form of the vice reflector bracket to improve the stiffness.

3.2. Wind Load
The wind load is one of the main loads for Antenna working in the open air in the strength calculation. As the wind speed changes irregularly over time, the wind load is dynamic. For the structure with better rigidity and higher quality, the wind doesn’t cause large vibration to the structure and it is a static effort on structure. Therefore, the wind load is treated as a static load in this paper.

The size calculation formula of the wind load is:

\[ F = \frac{1}{2} \mu_{Dn} \rho \bar{u}^2 A_n \]

- \( \mu_{Dn} \) — the wind pressure coefficient at the position of the block n unit;
- \( \rho \) — Air Density, take 1.225kg/m³;
- \( \bar{u} \) — Average wind speed (m/s);
- \( A_n \) — The characteristic area of block n unit (m²).

For loading the wind load, it is needed to generate the wind pressure file and then load the file into the Finite Element Model. As an APDL command flow file, wind pressure file includes command flow statements of pressuring force on each unit on the reflector panel. Before generating the wind pressure file, it is required to extract the information of nodes and units from the Finite Element Model of antenna structure. The wind pressure file statement style is:

```
SFE, 4000, 1, PRES, , 2.355828221E-004
SFE, 4001, 1, PRES, , 2.355828221E-004
```

With a full constraint to the lowest-ring lower hanging node of all radiating beams in the case of self-weight, adds Steady-state wind loads of 16m/s, 20m/s and 25m/s respectively; When the antenna is in pointing flat, calculates the deformation of each direction under three kinds of operating conditions; the maximum displacement in the X, Y, Z three directions are shown in table 1. From the results of calculation, it can be analyzed whether the deformation of antenna meets the design requirements or not.

| Direction   | Working Condition | Self-weight +16m/s | Self-weight +20m/s | Self-weight +25m/s |
|-------------|-------------------|--------------------|--------------------|--------------------|
| Max X displacement | 5.936mm      | 9.212 mm          | 14.332 mm          |
| Max Y displacement   | -7.635 mm     | -10.912 mm        | -16.031 mm         |
| Max Z displacement   | 9.927 mm      | 14.28 mm          | 21.083 mm          |

4. Conclusion
With APDL and MATLAB, this paper is achieved to establish the structure model of reflector antenna. In the process of modeling, the rules of numbering the key points, lines, nodes and so on are employed effectively and the modularization of modeling is achieved, which provides great convenience for the subsequent model changes. Meanwhile, taking the load of self-weight, wind load into consideration, the paper makes an analysis on the Finite Element Statics on the model in parameterization method. Establishing the structure model of reflector antenna not only verifies the correctness of model but also provides a basis for the optimization of antenna structure.

References
[1] Zhao, F.L. (2015) Structural Design and Mechanical Performance Analysis of Umbrella-type Deployable Antenna System. Shanghai Jiao Tong University.
[2] Yang,Y., Zhang D.L., Liu X.J. (2018) A Method for Calculating Translucent Shadow of Spacecraft Large Mesh Antenna, Journal of Astronautics, 39:366-382.
[3] LI Q., Zhu M.B. (2010) Thermal deformation analysis of large deployable truss antenna on satellite. Computer Aided Engineering, 19:40-43.

[4] Feng ZH.G., ZHeng Y.P. (2008) Establishment Method of a Finite Element Model of 50m Antenna Structure. Radio Communications Technology, 34:26-27,47.

[5] Gong SH.G., Huang Y.Q. (2009) Finite Element Analysis, ANSYS APDL Programming and Advanced Application. Machinery industry press, Beijing.