

Finite Element analysis for stress analysis and Non-linear contact analysis of Multi range load cell in ANSYS software

Shreyas Pandit¹, Prof. Vijay Kamble²

¹ PG student, Department of Mechanical Engineering, Textile and Engineering Institute, Ichalkaranji, India.

² Professor, Department of Mechanical Engineering, Textile and Engineering Institute, Ichalkaranji, India.

Abstract - Contact problems are more often nonlinear and require significant computer resources to solve. It is important that you know about the physics of the problem and take the time to set up your model to run as efficiently as possible. Contact problems present two significant difficulties. First, we generally do not know the regions where contact starts until problem is simulated. Depending on the loads, material, boundary conditions, and other factors, surfaces can come into and go out of contact with each other can't be predicted easily. Second, most contact problems undergoes friction. There are many of friction laws and problems to choose from which mainly are nonlinear. Frictional response of such problems can be confusing, making solution convergence difficult. This paper provides information on non-linear contact analysis with an illustration of "Multi-range load cell" in ANSYS software various tools required for analysis are also given but many times differs according to problem physics.

Key Words: Non-linear contact analysis, load cell, FEA, Ansys.

1. INTRODUCTION

Most of the contact problems in practice are of Non-linear in nature. There is a need to analyze that problem in analysis software packages with giving certain essential commands and inputs to specific problems according to its nature and geometry. Some of the basic cases include two parallel cylinders, two spheres, cylinders on a flat plate, gear teeth, roller bearings, etc. Theories and formulas for these cases are discussed in standard design books. [1, 2]. Factors that are to be taken into considerations are optimal mesh size, boundary conditions, loading, connections commands

This paper illustrates Multi range load cell which has small gap between contacting elements. Curved surface contact with horizontal flat surface in non-linear manner as the load goes on increasing.

While carrying FEA, care must be taken for giving inputs in connection table.

Multi-stage load cell is another modification over standard load cells based of strain gauge.

In the first range for light loading, there was no contact between the two special components of the load cell. In the second range for heavy loading, contact between the two components occurred.

Difference contact elements are as follow:

- Point-to-point gap elements
- Point-to-line (or slide-line) contact elements
- Point-to-surface contact elements
- Surface-to-surface contact elements [3]

For this case Surface-to-surface contact elements are chosen and steps on contact analysis are discussed below in next section.

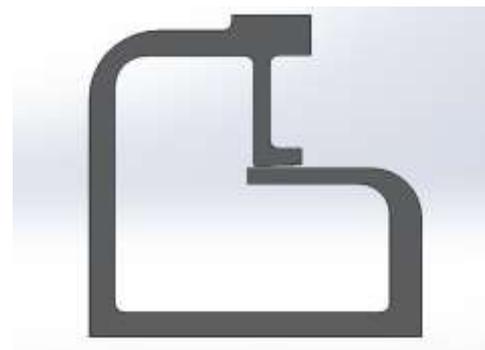


Fig - 1: Multi range Load cell

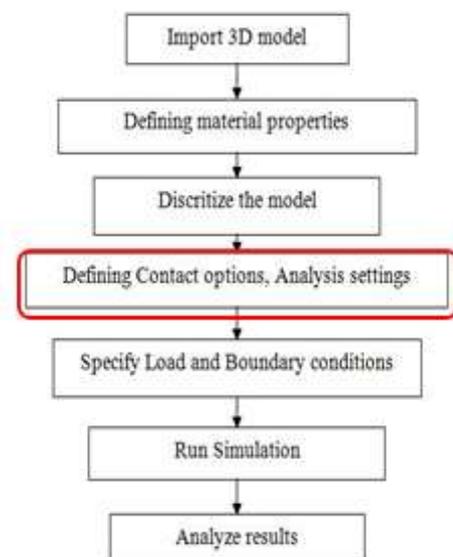


Fig 2:- Steps in FEA

Fig 2 shows steps steps involved in general finite element analysis, while the step indicated by red colour shows the topic of current interest i.e, contact analysis.

2. NON-LINEAR CONTACT ANALYSIS SETTINGS

There are number of settings that are required to be carried out so that after analysis solution converges as expected. Some of the contact settings are described in brief below.

The ANSYS FEA software supports three types of contact models: point-to-point, point-to-surface, and surface-to-surface.

• Steps in a Contact Analysis

The basic steps for performing a typical surface-to-surface contact analysis are listed below. Each step is then explained in detail in the following pages.

1. Create the model geometry and mesh.
2. Identify the contact pairs.
3. Choose contact and target surfaces.
4. Define the target surface.
5. Define the contact surface.
6. Set the element key options and real constants.
7. Define/control the motion of the target surface (rigid-to-flexible only).
8. Apply necessary boundary conditions.
9. Define solution options and load steps.
10. Solve the contact problem.
11. Review the results.

These steps are discussed below with brief of every stage in process,

1. First step is to mention contact method which may be:

- Bonded type contact
- Non separation type contact
- Frictional contact
- Frictionless contact
- Forced Frictional contact
- Rough contact

At the same time, we have to select the contact pairs which are usually called:

- Contact bodies.
- Target bodies.

2. Next step is selection of contact formulation:

The penalty method uses a contact “spring” to establish a relationship between two contacting surfaces. LaGrange method usually is considered for better conditioning and is less sensitive to the magnitude of the contact stiffness, compared to the penalty method [4,5]. However, in some special cases, the augmented LaGrange method is required for analysis with additional iterations, eg. especially if the deformed mesh gets too distorted [6].

3. Detection method and symmetry definition Contact detection is made at

- Gauss Points.
 - This contact detection is by default in ANSYS software.
 - Shows error at corner points.
- Nodal Points.
 - This detection method reduces convergence time.
 - They show instability for certain shapes like L shape.
- Nodal Projection from contact.

One more decision is to select an asymmetric or symmetric contact. The asymmetric contact is defined as having all contact elements on one surface and all target elements on the other surface. This is usually the most effective method to model the surface-to-surface contact problem [7,8].

However, under some conditions the asymmetric contact does not perform satisfactorily, i.e. each surface to be both a target and a contact surface. That means generation of two sets of contact pairs between the contacting surfaces. This is known as the symmetric contact.

4. Contact Stiffness and Penetration

Default stiffness value in ANSYS workbench is 1. The use of 1.0 value for stiffness is recommended for general bulk deformation dominated problems. For bending-dominated situations, a smaller value of 0.1 may be useful if convergence difficulties are encountered. If the contact

stiffness (which is by default condition) is too large, the model may oscillate, with contacting surfaces bouncing off of each other, leading problems so it is essential to keep the option of "Updating stiffness each iteration" ON to take into consideration increasing loading condition and of factors.

The lower the contact stiffness factor, the higher the penetration. However, because of this solution convergence often becomes faster/easier.

5. Pinball Region

A circular or spherical region near each contact element or target element is used to make pinball region determination. ANSYS has predefined values for the size of this pinball region, but the user can use his own values instead of the default to set specific levels for this parameter. Much amount of computation time is spent on contact elements in-contact or near-contact, as it is the area of interest than on the elements determined far-from-contact based on the pinball size.

6. Other settings that affect result are:

- Mesh Element
 - Selecting proper mesh element according to geometry is necessary.
 - Smaller mesh element size and use of tools like refinement and inflation helps in giving good results

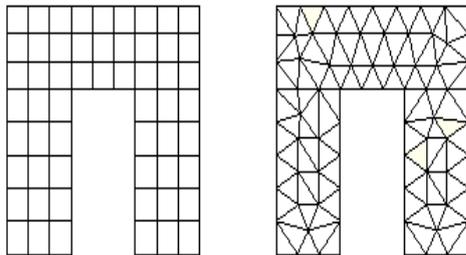


Fig - 2: A model meshed with quadrilateral elements and triangular elements

- Solver control:
 - Number of Loading steps/ sub steps (Analysis settings).
 - More sub steps helps in getting better convergence.
 - Also solver control tools like large deflection method can be used.
- Solution by Direct or iterative method.

3. NON-LINEAR CONTACT ANALYSIS FOR MULTI RANGE LOAD CELL

1. Contact Method

For Multi range load cell contact method selected was Frictionless. As the contacting surfaces come into contact, stiffness keeps on adding exponentially. Contact surface and Target surface for contacting pair.

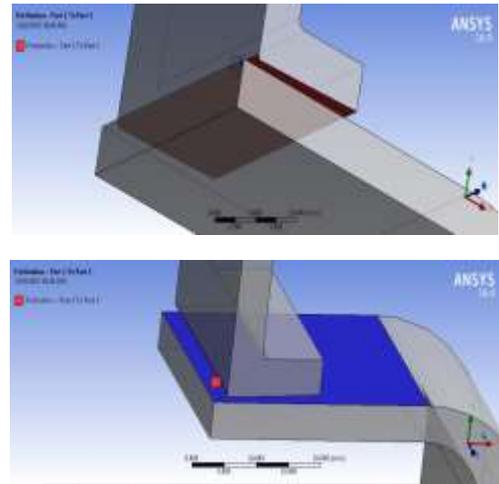


Fig - 3: Contact and target bodies

2. Contact Formulation.

As this load cell works on large deflection principle, it will be good to use augmented LaGrange method, instead of penalty or Gauss methods.

Definition	
Type	Frictionless
Scope Mode	Manual
Behavior	Asymmetric
Trim Contact	Program Controlled
Suppressed	No
Advanced	
Formulation	Augmented Lagrange
Detection Method	Nodal Normal To Target
Penetration Tolerance	Program Controlled
Normal Stiffness	Manual
Normal Stiffness Factor	1.e-002
Update Stiffness	Each Iteration
Stabilization Damping Factor	0.
Pinball Region	Radius
Pinball Radius	0.25 mm
Time Step Controls	None
Geometric Modification	
Interface Treatment	Adjust to Touch
Contact Geometry Correction	None
Target Geometry Correction	None

Fig - 6: Contact Settings

3. Detection method and symmetry definition

To get lesser convergence time and better plots Nodal normal to target option is selected.

As the geometry is having all contact elements on one surface and all target elements on the other surface, asymmetric contact is selected.

4. Contact Stiffness and Penetration

In this load cell, there is bending of the upper and the lower horizontal elements when the load is added in succession. Hence stiffness of the load cell is manually entered 0.01 to get good convergence plot. Also option of update stiffness on each iteration is kept ON.

For this case, penetration is kept “program controlled”.

5. Pinball Region

Pinball radius for this case is put manually, large pinball radius size may have considerable effect on output results.

It is shown in fig.5 below, (blue sphere)

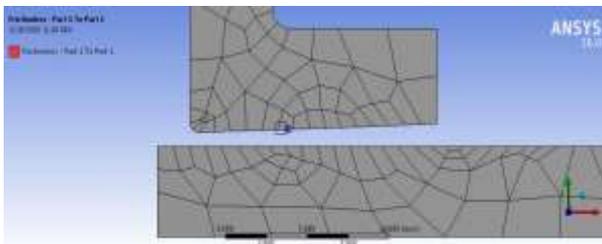


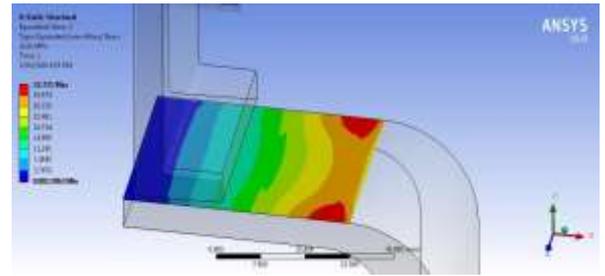
Fig - 5: Pinball

6. Other settings that affect result are:

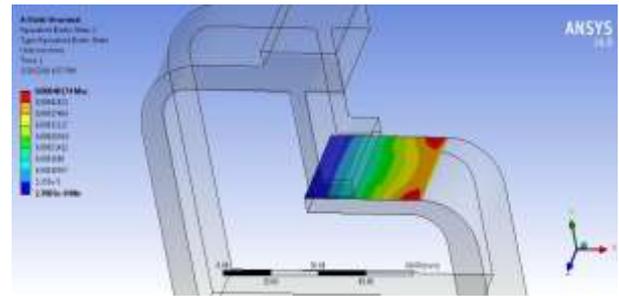
- Mesh Element
 - Density of mesh elements is taken medium
 - Element size 1 is taken at contact pairs
- Solver control
 - To attempt effective loading maximum 20 sub steps are provided.
 - Direct solver is enough for this type of problems to get convergence.
 - Large Deflection is kept ON, while weak spring deflection option is kept OFF

I. Results

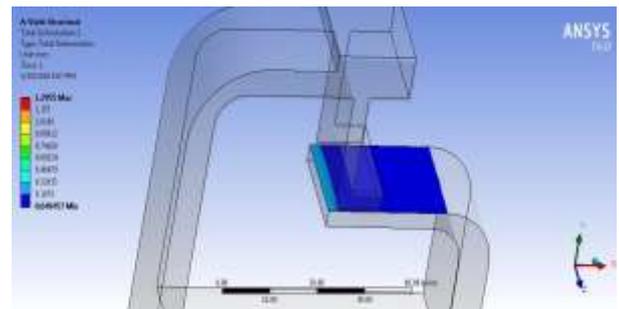
After giving inputs to preprocessor of ANSYS, result are evaluated to get contact results.



(a) Stress results



(b) Strain results



(c) Displacement results

Fig - 7: Results

Fig 7 shows results obtained at the output showing stress(a),strain(b),and displacement(c) at the contacting region.

Stress and strain are shown maximum at the curve end portion of the middle portion of load cell as it will act as cantilever beam.

Also, Displacements is maximum near the first contacting region as displacement first starts at that point and continuously progress as load.

4. CONCLUSIONS

Analysis package ANSYS offers various tools to deal with contact problems. Some of these tools were discussed in the paper by giving a specific example “Multi range load cell”. These tools include specifying contact type, contact and target surfaces, contact formulation and detection, stiffness and pinball radius etc. As problem change specifications to be provided are to be changed taking into account for physics of problem.

REFERENCES

- [1] Shigley, J. E., Mechanical Engineering Design, second edition, McGraw-Hill, Eighth Edition, pp. 117-120.
- [2] Juvinall, R. C. and K. M. Marshek, Fundamentals of Machine Component Design, second edition, John Wiley and Sons, pp. 322-329.
- [3] David H. Johnson, P.E. "Principles of simulating contact between parts using ANSYS", Penn State-Erie Erie, Pennsylvania, USA.
- [4] J. E. Mottershead, S. K. Pascoe and R. G. English, "A General Finite Element Approach For Contact Stress Analysis", International Journal For Numerical Methods In Engineering, Vol. 33, 765-779 (1992)
- [5] Grama R. Bhashyam, "ANSYS Mechanical- A Powerful Non-linear Simulation tool", pp 20-21, 30-34.
- [6] Anil B. Chaudhary, Klaus-Jurgen Bathe4 Solution Method For Static And Dynamic Analysis Of Three-Dimensional Contact Problems With Friction, Compwers & Strwrwes Vol. 24. No. 6. Pp. M-873. 1986.
- [7]http://www.ansys.stuba.sk/html/guide_55/g-str/GSTR9.htm
- [8] Ivana Atanasovska, Vera Nikolić-Stanojlović, Dejan Dimitrijević, Dejan Momčilović; "Finite Element Model for Stress Analysis and Nonlinear Contact Analysis of Helical Gears", Scientific Technical Review, Vol.L VIX,No.1,2009, pp. 63-65.