Mesh convergence study and estimation of discretization error of hub in clutch disc with integration of ANSYS

Hemesh Patil1 and P V Jeyakarthikeyan*2

1Student, Department of Mechanical Engineering, SRM Institute of Science and Technology, India.
2Assistant Professor (Selection Grade), Department of Mechanical Engineering, SRM Institute of Science and Technology, India.

*Corresponding author: jeyapv@gmail.com

Abstract. Structural analysis is an essential tool for design engineers. Mesh generation is the basic step in any simulation. In practice of finite-element stress analysis, the engineer first needs to know if key stresses are converging, and second if they have converged to a reasonable level of accuracy. In order to achieve results that are reliable when using the finite element method, it must be ensured that an acceptable mesh is used with respect to the shape and size of the elements. Mesh quality and mesh density values are directly linked with the solution accuracy. This paper presents a mesh convergence study on the hub which is the most challenging part in the automotive clutch disc assembly. Finite element analysis convergence defines the relationship between the number of elements or DOF and the analysis accuracy. Discretization error is key concept for the study of mesh convergence study, also discretization is function of number of degree of freedom of model. A detailed comparison of different results is carried out using convergence test based on displacement and energy norms. Error between exact and ANSYS results are compared in this study.

1. Introduction

Nowadays, Finite Element Analysis (FEA) is widely used in analysis since it has many advantages compared to the traditional analytical method. Also, there are many problems that cannot be solved by analytical methods but can only be solved by FEA. FEA is preferred for its dual advantages of reduction of the cost as well as faster designing of any component. It gives a typical two-way solution to the question, “If any change in design is made then what?”. FEA is a numerical technique to find the approximate solution of partial differential equations of stress analysis due to the impact of force, vibration, heat etc.

FEA is extensively used in many fields one of which is the automobile industry. In this industry, it is primarily used in power transmission systems. The main objective of a power transmission system is to transfer efficient amount of power to the wheels from the engine. Researchers continue to focus on an optimal design of the power transmission system. Hub is used as a power transmitting part and is present in the clutch disc assembly. Frictional pads transmit power from the flywheel to the hub in the clutch plate and
from there to the output shaft. Hub also acts as a torque absorption unit, it absorbs the torque at the time of engagement and disengagement of clutch disc with the help of springs located in the drive plate.

Nodal stress result and interpolated recovered results at node points are studied with post processing error estimation by Zienkiewicz and Zhu[1]. Later Hyung-Seok and Batra R C studied Zhu error for p refinement of mesh [2]. It is very important for researchers to perform a mesh convergence study to obtain the required meshing size by creating a model since this can result in large variations to the output [3]. In case of mesh convergence study in ANSYS, Shah Chandresh presented techniques to evaluate mesh convergence errors [4]. The mesh convergence errors are allows to monitor whether the numerical solution is sufficiently accurate even if the exact solution is unknown [5]. Aspect ratio, skewness and maximum corner angle plays an important role in analysis [6]. Devals C, Vu T C, Zhang Y, Dompierre J and Guibault F studied mesh convergence study for turbulent flow in hydraulic turbine draft tubes [7].

1.1. Mesh discretization error

Displacement is the unknown variable in any FEA, displacement is calculated at every nodes in the model. Stresses are obtained by first derivative of displacement field, and is got from Equation 1,

\[ [F] = [K][u] \]  

where \([K] = \text{Global Stiffness matrix}\)  
\([u] = \text{Displacement Vector}\)  
\([F] = \text{Force Vector}\)

Stress values from ANSYS are an average of stresses from all elements attached to that node. This introduces the error in magnitude of the stress value, figure 1 shows the mesh discretization error. As the mesh is coarse, there is greater difference between stress values of the adjacent elements.

![Figure 1. Mesh Discretization Error](image)
change in stress value even after the refinement in mesh. For this reason, the analyst should first check the discretization errors in the FE solution.

1.2. ANSYS error estimation
The errors presented below have been explained by Shah Chandresh [4]

- Percentage Error Energy Norm (SEPC): SEPC is a rough estimate of the stress error over the entire set of selected elements (obtained from prerr command in ANSYS APDL).
- Element Stress Deviations (SDSG): SDSG is to measure the difference between elemental and nodal stress. The difference between averaged and unaveraged stress gives an idea about mesh density (obtained from presol and prnsol command in ANSYS APDL).
- Element Energy Error (SERR): SERR is the energy associated with the stress mismatches of all the nodes of the element (obtained from presol and prnsol command in ANSYS APDL).
- Maximum and Minimum Stress Bounds (SMXB, SMNB): The stress bounds are used to determine the effect of mesh discretization error on the maximum stress (obtained from stress plots plsol command in ANSYS APDL).

Exact solution can also be calculated using richardson’s extrapolation and gauss quadrature [8], FE solution will converge towards the exact solution with increasing number of elements or with increasing order of polynomial in the element. If the following requirements are fulfilled, then the solution is converged. The element must be able to represent constant strain which is possible where small elements are used. Small elements have strains are close to constant over the elements. In order to avoid the gaps, adjacent elements must be compatible i.e. the connecting nodes have the same possibility to move as concerned element nodes.

2. Finite element accuracy criteria
Four types are used in this study of mesh discretization error analysis; they are used to calculate global as well as local mesh discretization errors.

- Type 1A (SEPC_Model): The error norm of the entire model must be less than 15%.
- Type 1B (SEPC_Local): 3 layers of elements are selected as local area at high stress, the error norm of the local area of high stress must be less than 10%.
- Type 2 (Coefficient of variation): In local area of high stress, the average stress value of local elements has coefficient of variation stress less than 5%. COV (Coefficient of variation) is calculated as ratio of nodal stress to the mean stress value.
- Type 3 (%Error): The difference between the stress component and stress considering the discretization in local area of high stress must be less than 7%.

\[
\text{%Error} = \frac{\text{SMXB} - \text{SMX}}{\text{SMX}} \times 100
\]

\[
\text{SMXB} = \text{Stress considering the discretization error (Calculated by ANSYS APDL)}
\]

\[
\text{SMX} = \text{max stress value in selected set of element.}
\]

- Type 4A (RMS_Model): This type uses the absolute value of SDSG (maximum variation of nodal component) to \(S_{\text{eq}}\) (Von - Mises stress) at that element and RMS (Root mean square) is calculated from that values and it should be less than 15%.
- Type 4B (RMS_Local): (SDSG/ \(S_{\text{eq}}\)) RMS value of local area having high stress should be less than 10%.
These types are implemented and ACT (ANSYS Customization Tool) is developed with integration of xml and python to calculate the above results automatically. figure 2 shows graphical user interface of the ACT in ANSYS Workbench.

![Figure 2. Ansys ACT](image)

## 3. Example problem and results

Hub in clutch disc assembly is used to demonstrate the mesh convergence criteria and discretization errors. The model is developed for the study is simulated using ansys workbench for static structural analysis. Figure 3 shows the damper hub model with inner diameter of 20mm and outer diameter of 50mm having fillet of 0.5mm each at critical region. The material used for the hub is 42CrMoS4 having young’s modulus (E) of 210000 Mpa and poisson’s ratio (v) of 0.3.

![Figure 3. Geometric model of hub in clutch disc assembly](image)
Boundary condition for the analysis is given as over torque of 600 Nm is applied on the outer splines of the hub, Inner and outer splines are locked in all DOF except rotation about z axis for outer splines and three equally spaced nodes are locked in z direction for convergence stability.

In this study, solid hexahedron elements are used for meshing as hexahedron elements give better accuracy compared to tetrahedron elements. Different meshes are developed with different body, face, edge sizing control in ansys workbench. Three different meshes and their maximum principal stress plot within iterations are showed in figure 5 to figure 10 respectively. Details for the mesh are as given below in table 1 for iteration 1, iteration 2 and iteration 3.

![Figure 4. Applied boundary conditions](image)

| Iteration No | Number of elements |
|--------------|--------------------|
|              | Inner spline      | Outer spline |
|              | Across length     | Across fillet | Across length | Across fillet |
| 1            | 40                | 3            | 10           | 3            |
| 2            | 50                | 5            | 15           | 5            |
| 3 (with inflation) | 60              | 5            | 20           | 5            |

Table 1. Mesh details for iteration 1, iteration 2 and iteration 3
Figure 5. Mesh for iteration 1

Figure 6. Mesh for iteration 2
Figure 7. Mesh for iteration 3 (inflation)

Figure 8. Max principal stress plot for iteration 1
Figure 9. Max principal stress plot for iteration 2

Figure 10. Max principal stress plot for iteration 3 (inflation)
Results for each type are tabulated in table 2 below for inner and outer spline of the hub.

### Table 2. APDL based results for the hub model for different iterations

| Type of element | No of elements Across Length | DOF | Max Principal Stress (Mpa) | % Error | SEPC Model | SEPC Local | Root Mean Square Model | Root Mean Square Local | COV |
|-----------------|-----------------------------|-----|-----------------------------|---------|------------|------------|------------------------|------------------------|-----|
| Inner Spline    |                             |     |                             |         |            |            |                        |                        |     |
| L.O.            | 40                          | 3   | 120                         | 737.94  | 31.71      | 18.29      | 11.08                  | 48.38                  | 31.69 | 1.71 |
|                 | 50                          | 3   | 150                         | 736.19  | 29.07      | 18.35      | 10.73                  | 48.41                  | 31.84 | 1.71 |
|                 | 60                          | 3   | 180                         | 739.82  | 28.91      | 17.42      | 10.48                  | 48.42                  | 31.81 | 1.71 |
|                 | 50                          | 5   | 250                         | 865.86  | 18.87      | 13.25      | 9.59                   | 32.74                  | 18.61 | 1.55 |
| H.O.            | 50                          | 3   | 300                         | 900.60  | 11.93      | 13.30      | 8.34                   | 13.58                  | 7.40  | 3.65 |
|                 | 60                          | 3   | 360                         | 916.20  | 9.26       | 9.41       | 7.28                   | 13.67                  | 7.39  | 3.65 |
|                 | 40                          | 5   | 400                         | 929.12  | 9.50       | 8.82       | 4.86                   | 7.63                   | 4.55  | 3.53 |
| Outer Spline    |                             |     |                             |         |            |            |                        |                        |     |     |
| L.O.            | 10                          | 3   | 30                          | 602.50  | 13.89      | 10.92      | 15.75                  | 22.44                  | 10.78 | 1.44 |
|                 | 15                          | 3   | 45                          | 631.96  | 10.65      | 10.73      | 17.16                  | 24.63                  | 13.04 | 1.27 |
|                 | 10                          | 5   | 50                          | 653.03  | 13.75      | 10.18      | 16.65                  | 18.59                  | 11.47 | 1.06 |
| H.O.            | 10                          | 3   | 60                          | 685.35  | 11.90      | 5.51       | 6.65                   | 5.24                   | 3.86  | 2.31 |
|                 | 15                          | 3   | 75                          | 712.89  | 11.50      | 9.85       | 15.41                  | 16.74                  | 10.08 | 0.95 |
|                 | 20                          | 5   | 100                         | 739.57  | 7.01       | 9.48       | 16.26                  | 15.46                  | 9.01  | 0.91 |
| H.O.            | 20                          | 3   | 120                         | 753.38  | 3.78       | 7.28       | 4.67                   | 4.10                   | 2.36  | 2.94 |
|                 | 15                          | 5   | 150                         | 759.58  | 3.18       | 8.07       | 5.66                   | 3.37                   | 2.26  | 2.25 |
|                 | 20                          | 5   | 200                         | 761.95  | 2.60       | 6.87       | 3.93                   | 2.59                   | 1.81  | 2.29 |
| Outer Spline    |                             |     |                             |         |            |            |                        |                        |     |     |
| L.O.            | 10                          | 3   | 30                          | 606.96  | 11.61      | 14.73      | 10.36                  | 16.29                  | 8.94  | 1.27 |
|                 | 15                          | 3   | 45                          | 608.69  | 11.57      | 15.71      | 10.60                  | 16.07                  | 8.70  | 1.27 |
|                 | 10                          | 5   | 50                          | 610.02  | 11.53      | 15.08      | 10.92                  | 16.39                  | 8.74  | 1.27 |
| H.O.            | 10                          | 3   | 60                          | 642.87  | 10.72      | 24.77      | 8.36                   | 19.44                  | 9.95  | 2.85 |
|                 | 15                          | 5   | 75                          | 648.61  | 9.72       | 26.18      | 5.51                   | 20.35                  | 10.48 | 2.83 |
|                 | 20                          | 5   | 100                         | 652.73  | 9.55       | 21.28      | 8.03                   | 21.91                  | 15.54 | 2.65 |
| H.O.            | 20                          | 3   | 120                         | 655.52  | 8.90       | 12.34      | 9.85                   | 12.58                  | 7.75  | 1.09 |
|                 | 15                          | 5   | 150                         | 656.67  | 6.94       | 12.63      | 10.01                  | 13.12                  | 7.91  | 1.10 |
|                 | 20                          | 5   | 200                         | 658.00  | 6.27       | 12.24      | 9.48                   | 13.38                  | 7.80  | 1.09 |

*L.O. = Lower Order, H.O = Higher Order, DOF = Degree of Freedom
The graph of degree of freedoms in higher stress region vs. maximum principal stress can be plotted as shown in figure 11.

**Figure 11.** Graph of DOF vs maximum principal stress for inner and outer spline
4. Discussion
Different iteration is carried out by changing the number of elements at the high stress region that is at fillet region, in inner spline and outer splines as the number of elements is keep on increasing the stress value is getting more and more accurate. By seeing in table 2 in between many iteration stress values are converging where the percentage of error is less than 7%, as well as other criteria is satisfying in the same as discussed in section 2.

5. Conclusion
From the above graphs figure 11, it can be seen that the results are converging even after refining the mesh parameters, but as the mesh is refined the time required for the simulation also increases. Therefore, this mesh convergence study gives a fair idea of whether the solution converged or not. From table 2, the following conclusions are made percentage error value must be less than 7%, Error norm (SEPC) for model must be less than 15% and for the local region it must be less than 10%, Coefficient of variation must be less than 5% and Root mean square value must be less than 15% for the model and 10% for the local region.

Study gives idea about importance of mesh convergence and methods of estimation of discretization error. This will help analyst to understand the FE results of his solution more accurately. The FEA results are significantly affected by the meshing techniques used into the analysis as well as it depends on the element type and size of the element. A number of elements are more general than the size of element at the time of meshing as the length of the element is more dependent on the part being meshed. Mesh convergence study, helps to find the accuracy of the FEA solution. The method discussed above gives a fair idea about the importance of mesh convergence and method of error of estimation.

6. Reference

[1] Zienkiewicz O C and Zhu J Z 1987 A simple error estimator and adaptive procedure for practical engineering analysis, International journal for numerical methods in engineering. 24, 337-357
[2] Oh H S and Batra R C 1999 Application of Zienkiewicz–Zhu's error estimate with super convergent patch recovery to hierarchical p-refinement, Finite elements in analysis and design. 31, 273-280
[3] Ahmad M, Ismail K A and Mat F 2013 Convergence of Finite Element Model for Crushing of a Conical Thin-walled Tube, Procedia Engineering. 53, 586-593
[4] Shah C 2002 Mesh discretization error and criteria for accuracy of finite element solutions, In Ansys Users Conference, 12
[5] Bespalov A, Haberl A and Praetorius D 2017 Adaptive FEM with coarse initial mesh guarantees optimal convergence rates for compactly perturbed elliptic problems, Computer Methods in Applied Mechanics and Engineering. 317, 318-340
[6] Vassberg J, DeHaan M and Sclafani T 2003 Grid generation requirements for accurate drag predictions based on OVERFLOW calculations, In 16th AIAA computational fluid dynamics conference, 4124
[7] Devals C, Vu T C, Zhang Y, Dompierrre J and Guibault F 2016 Mesh convergence study for hydraulic turbine draft-tube, In IOP Conference Series: Earth and Environmental Science. 49, 082021
[8] Jeyakarthikeyan P V, Yogeshwaran R and Abdullahi H S 2016 Time efficiency and error estimation in generating element stiffness matrices of plane triangular elements using Universal Matrix Method and Gauss-Quadrature, Ain Shams Engineering Journal.