॥ सा विद्या या विमुक्तये ॥
भारतीय प्रौद्योगिकी संस्थान धारवाड
Indian Institute of Technology Dharwad

PREDICTION OF SCF USING DEEP LEARNING

PROJECT REPORT

ME 324: Design of Machine Elements.

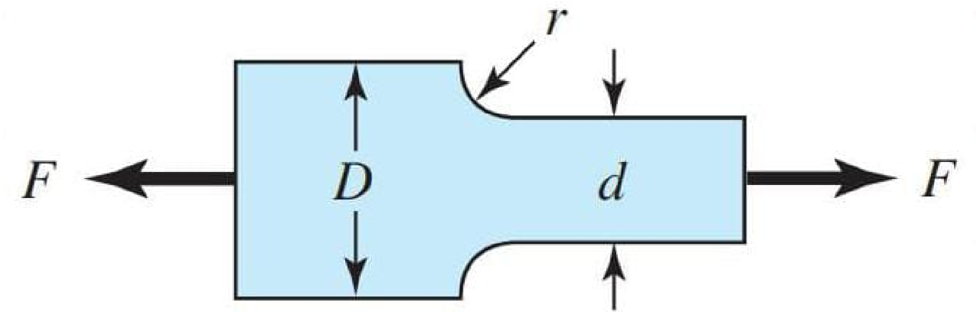
Name: *K. Vijay Prabhas-200030028. D.Dhanush-200030018*

Instructor:	*Tejas Prakash Gotkhindi*
Department:	Mechanical, Materials and Aerospace Engineering (MMAE)
Date:	November 9, 2022

Contents

1 PROBLEM STATEMENT

Finding the stress concentration factor of a thick bar with a fillet is loaded axially with force using the deep learning concept.

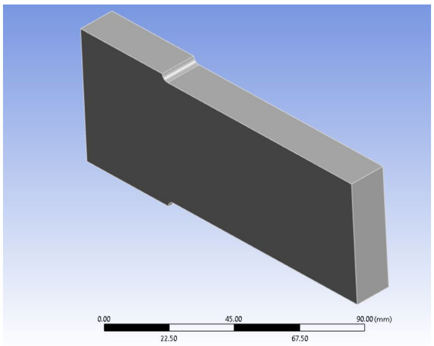


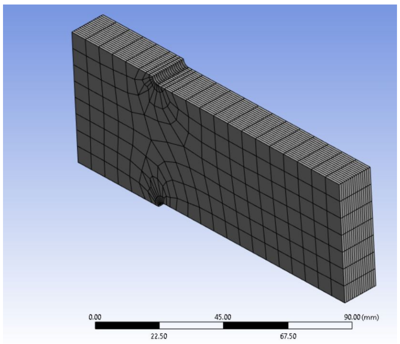
2 ANSYS MODEL

About Ansys: ANSYS is a general-purpose, finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis, heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

we can run different applications from Ansys that we can choose from the toolbox, based on our need we can choose the required system we need to proceed the problem.
in our case we choose static structural from the toolbox.

- By selecting the static structural the model is designed in the Design Modeler software.
- The material of the specimen is structural steel.
- Going to Ansys mechanical in the project schematic. Initially, the mesh is generated to the specimen, which is the default. The sizing in the mesh is applied to increase the accuracy, particularly at the stress concentration region.
- Sizing in the mesh is generally used for refining of the complete mesh so that to get very good accuracy, we should choose an optimum value for the element size particularly for some regions where the stress concentration is high.
- so that we get almost exact stresses at that region.
- very small meshing may take huge time in order to simulate the model.

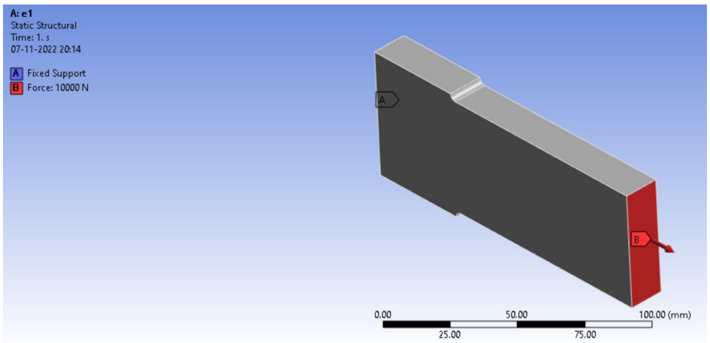




Boundary Conditions are very important in Ansys as we are much interested to solve our problem for the given specific conditions.

- Inside the Ansys the governing equations will run in order to solve for the applied boundary conditions.
- we can apply as many conditions we want.

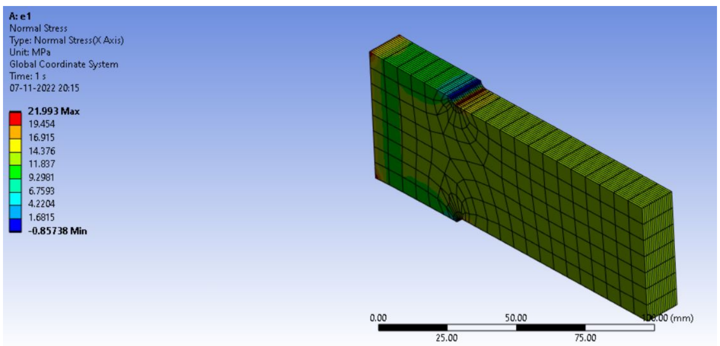
For this case the boundary conditions are, the fixed support was at the left and the force (e.g.,



10KN) at the right end.

3 SOLUTION OF THE FIRST SIMULATION

Solution includes the stress, we need to insert the stress in solution for the whole body.

- By clicking on solve for the stress component we get the required simulated specimen with the stress distribution.
- 

- From the solution picture we can say that the area with red color emphasises that the stress concentration is very high at that region i.e. at the fillet.

- Because due to sudden change in the geometry the stress concentration increases.

Finding stress concentration factor A stress concentration factor (Kt) is a dimensionless factor that is used to quantify how concentrated the stress is in a mechanical part. It is defined as the ratio of the highest stress in the part compared to a nominal stress.

- After knowing the stress acting at the fillet region, which is maximum, we then calculate the stress concentration factor,

$$\text{Stress concentration factor(SCF)} = maximumstress/nominalstress$$

- for this case the nominal stress is ,

$$\text{nominal stress} = F/d \times t \text{ (in MPa)}$$

where 'F' is the force applied,
'd' is the width of the narrow part
't' is the thickness of the specimen.

4 AUTOMATED PARAMETRIC STRUCTURAL ANALYSIS

Parametric Analysis is the important feature we use in this whole project.

- For research type of works we need to simulate the same specimen just by altering some geometrical parameters.
- in order to simulate or run the software for huge tests, it is difficult to run or test specimen for every time.
- so by using parametric structural analysis we can parameterize some of the parameters which we want to change for different tests.
- For this case, the parameters are the radius of the fillet and the width of the broad part. ('r' and 'D').
- we took over 500 design points which is least number we need to give to deep learning to train the model.
- with the help of **response surface** in design explorer we enter the design of experiments we want to conduct based on the parameters we have given.
- we can insert the excel file in the design of experiments just by changing the properties to custom mode.
- At last the automation in parametric analysis starts.

5 PREPROCESSING OF DATA

- The data collected from ANSYS have radius of the fillet (r), width of the big section (D) and maximum stress.
- For deep learning model the learning rate will be higher when all the data is close enough, so it would be better if the model fed the ratio of radius of the fillet to the width of the small section, the ratio of the width of the big section to the width of the small section, and SCF.
- While doing the simulations in ANSYS width of the small section, Axial Force and thickness of the specimen are kept constant, so the nominal stress is constant.
- So, in the data provided by the ansys dividing the r column by d gives r/d, D column by d gives D/d and maximum stress by nominal stress (10Mpa) gives SCF.
- Feeding this updated data to the deep learning model eliminates the need for normalization or standardization scalers.

6 IMPLEMENTATION OF CODE

Libraries used:

- Pandas : a python package providing fast, flexible, and expressive data structures designed to make working with "relational" or "labeled" data both easy and intuitive
- Numpy : a python library used for working with arrays
- Keras : a high-level, deep learning API for implementing neural networks
- Scikit-learn : an open-source python library that implements a range of machine learning, pre-processing, cross-validation, and visualization algorithms using a unified interface
- Matplotlib : a comprehensive library for creating static, animated, and interactive visualizations in python

After importing all the required libraries, the csv file containing all the preprocessed data is loaded into a panda dataframe. The required inputs (r/d) and (D/d) are loaded into one dataframe and SCF into another dataframe. The data is then split into train data and test data with a test size of 20 percent and random state 20 using traintestsplit() method from the scikit learn.
After the completion of split of the data, the deep learning model is built using keras library. Sequential model is called and 3 hidden layers are created having 128, 64, 64 neurons respectively. The activation function used in these hidden layers was 'relu'. The loss function used is 'mean squared error', optimizer used is 'adam' and metrics used is 'mean absolute error'. The number of epochs for which the model was trained is 100.

After the model is trained with train data, the model is made to predict for the test data and the results are compared with the actual results needed. A 3d scatterplot is plotted to compare the predicted data and actual data using matplotlib. The mean squared error from the neural net was '0.0044118'. The mean absolute error from the neural net was '0.03991721'.

The model made to predict the SCF for some arbitrary value given by user: 'r/d = 0.06', 'D/d = 1.5', the predicted stress concentration factor by the model is '2.5974438'.

7 CHALLENGES FACED

In Ansys,

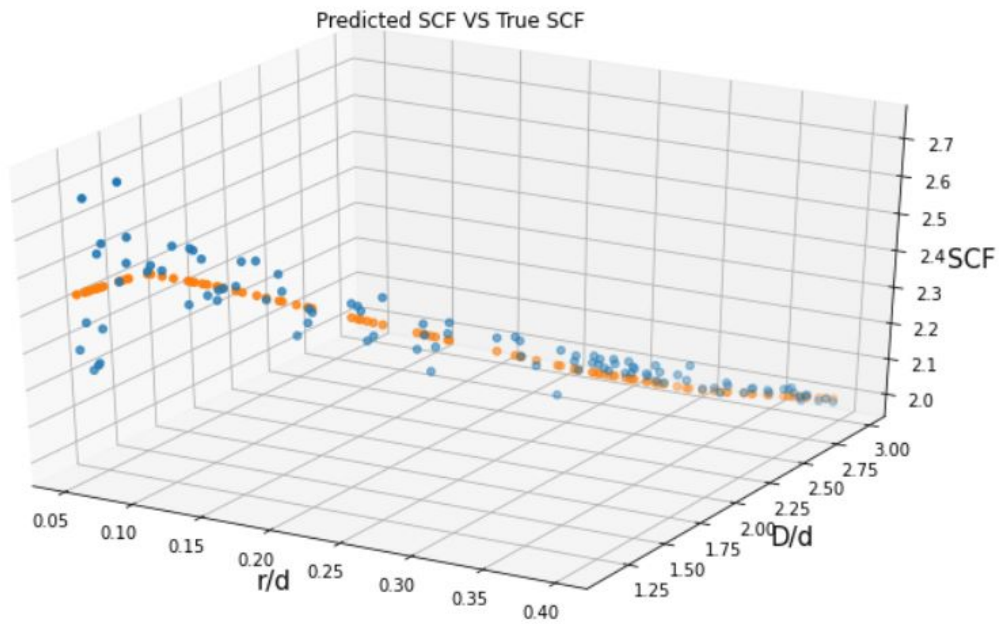
- First, we tried to design the model in the Space Claim software, but there is no option to select parameters in the Space Claim.
- For the parametric analysis part to generate the data for over 500 design points, it took more than 7 hrs.
- We also faced problems determining the parameter range to generate huge design points.

In Deep Learning,

- Increasing the number of hidden layers usually increases the accuracy of the model but after 3 hidden layers the accuracy decreased because of the overfitting.
- We tried increasing the number of nodes or neurons in each layer to achieve higher accuracy keeping the number of layers constant but after a certain number of nodes the accuracy decreased.

8 RESULTS

The plot of predicted vs true SCF for test data from deep learning,



For r = 3mm, D = 75mm, d = 50mm, F = 10kN, t = 20mm, the maximum stress obtained from the Ansys simulation was 26.501Mpa. So, SCF = 26.501/10 = 2.6501.

Table of Design Points

	A	B	C	D	E	F	G
1	Name	P1 - R	P2 - D	P3 - Normal Stress Maximum	Ret...	Retained Data	Note
2	Units	mm	mm	MPa			
3	DP 0 (Current)	2.5	55	21.993	☑	✓	
4	DP 1	3	150		☐		
5	DP 2	3	75	26.501	☐		Created from Design of Experiment
*					☐		

The SCF predicted by the deep learning model for the same is 2.5974438.

```
#input from user
import array as arr
rd = float(input("Enter the ratio of radius of fillet and width of the small section: "))
Dd = float(input("Enter the ratio of width of the big section and width of the small section: "))
i = [[rd,Dd]]
#print()
print("Stress Concentration Factor: ",model.predict(i)[0][0])
```

```
Enter the ratio of radius of fillet and width of the small section: 0.06
Enter the ratio of width of the big section and width of the small section: 1.5
1/1 [==============================] - 0s 68ms/step
Stress Concentration Factor:  2.5974438
```

ERROR CALCULATION,

relative error = (2.6501-2.5974438)/(2.6501) = 0.01987

percent error = 1.987%

Hence the percent error between actual SCF and the SCF predicted by the deep learning model is 1.987%.

9 REFERENCES

- https://youtu.be/55HyzZKtW6U
- https://youtu.be/EOWnMKCVHks
- https://www.analyticsvidhya.com/blog/2021/06/linear-regression-using-neural-networks/
- https://youtu.be/-vHQub0NXI4
- https://youtu.be/2yhLEx2FKoY