Universiti Malaysia PAHANG
Engineering • Technology • Creativity

Mechanics of Materials

Project 3 - 4

by
Dr Nanang Fatchurrohman
Faculty of Manufacturing Engineering
fatchurrohman@ump.edu.my

5.0 DISCUSSION

- In this part, we will be discussing on the structure of the I-Beam using Finite Element Analysis in Catia, in which the load is applied to the centre of it with respect to Y-axis that is 1962N.

- Below is the material properties for steel:

Material	Steel
Young's modulus	2e+011N_m2
Poisson's ratio	0.266
Density	7860kg_m3
Coefficient of thermal expansion	1.17e-005_Kdeg
Yield strength	2.5e+008N_m2

Figure 6: Material Properties for steel

- Below are the amount of mesh in the I-Beam:

Entity	Size
Nodes	12630
Elements	6326

Figure 7: I-Beam Mesh

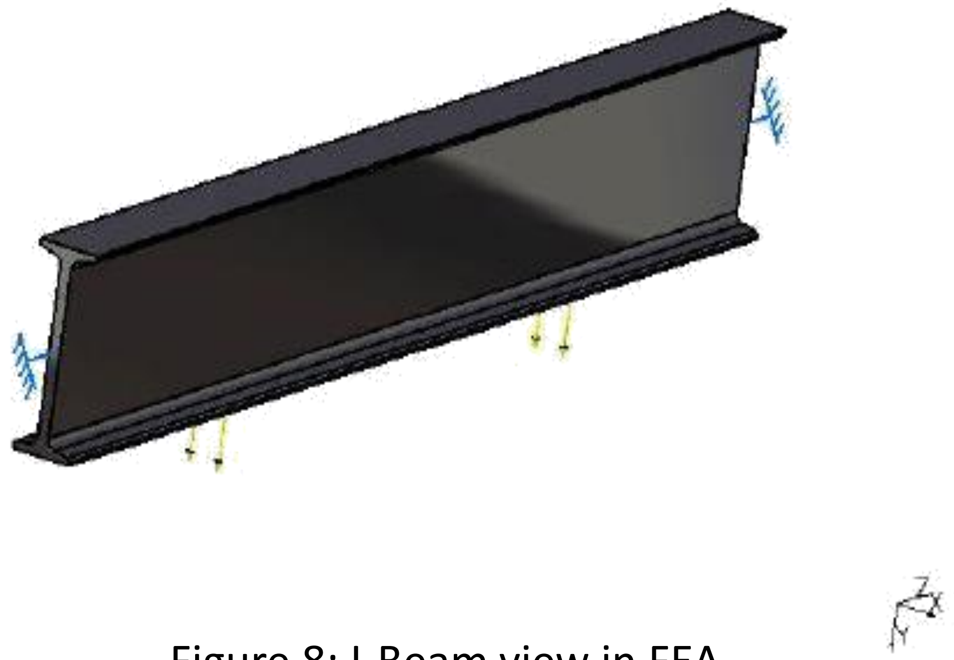

Figure 8: I-Beam view in FEA

- In this analysis, the result can be shown visually as from before the load is applied and after the load is applied. The figure below shows that the I-Beam in which the load is not yet being applied. The I-Beam is being is supported with the runaway beam and in the Finite Element Analysis we can consider the runaway beam as a clamp on each side of the I-Beam.

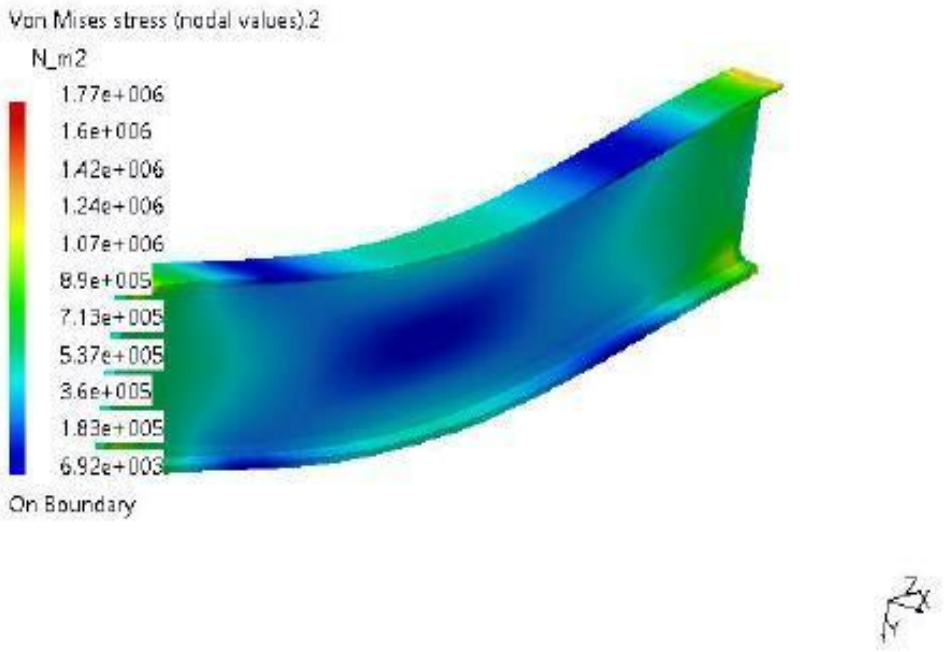

Figure 9: Von Mistress Analysis

- The above figure shows where the stress is most concentrated. Based on the Von Mises Stress, it is being visualised on which part of the beam having higher stress to the lowest stress. We can conclude that the centre of the beam is the most stressed area and the stress is decreasing towards the clamp on both of the end of the beam.
- Beam that have much higher stress tend to bend more from its more original shape which is most not preferably. However as it is not exceed the yield strength of steel which is 2.5e*008n_m2, the beam will not deform plastically in other words is permanently deform of its elastic limit. As an engineer it is important to prevent any plastic deformation to the structure because it will lead to fatal incident during operation probably.

6.0 CONCLUSION AND RECOMMENDATION

- Engineer had to analyse a simple structure theoretically, to prove its safe design, as any structure built requires to be tested for its structural integrity.
- Even though we have calculated using classical hand calculation, we have decided to test it using Finite Element Model (FEM) using CATIA as it is analysed with a greater precision than using the conventional hand analyses, because the actual shape, load and constraints, and material property combinations can be specified with much greater accuracy.
- Moreover Finite Element analysis (FEA) helps a lot in providing analysis data and deformation simulation to gives more understanding the process of deformation visually. The generated report also provide many information for calculation made. As an engineer, it is important to calculate their design structure to ensure safety of the design.

7.0 REFERENCES

1. -OVERHEAD CRANE- https://en.wikipedia.org/wiki/Overhead_crane
2. -OVERHEAD COMPONENT- http://www.totalcrane.com/bridge-crane-components.html
3. Mechanics of Materials, 6th Edition

Analysis1

MESH:

Entity	Size
Nodes	12630
Elements	6326

ELEMENT TYPE:

Connectivity	Statistics
TE10	6326 (100.00%)

ELEMENT QUALITY:

Criterion	Good	Poor	Bad	Worst	Average
Stretch	6326 (100.00%)	0 (0.00%)	0 (0.00%)	0.327	0.548
Aspect Ratio	3922 (62.00%)	2404 (38.00%)	0 (0.00%)	4.458	2.664

Materials.1

Material	Steel
Young's modulus	2e+011N_m2
Poisson's ratio	0.266
Density	7860kg_m3
Coefficient of thermal expansion	1.17e-005_Kdeg
Yield strength	2.5e+008N_m2

Static Case

- Boundary Conditions

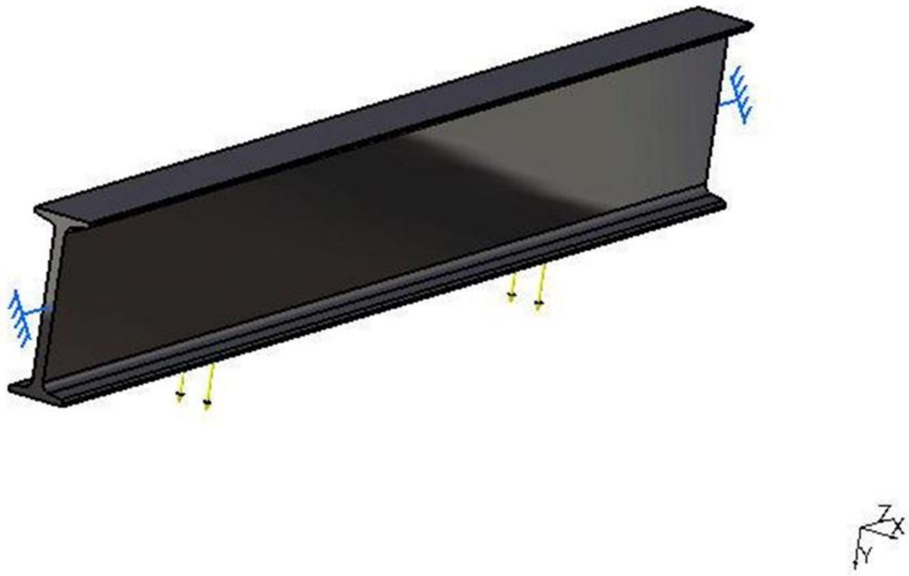

Figure 1

STRUCTURE Computation

Number of nodes	:	12630
Number of elements	:	6326
Number of D.O.F.	:	37890
Number of Contact relations	:	0
Number of Kinematic relations	:	0

Parabolic tetrahedron : 6326

RESTRAINT Computation

Name: Restraints.1

Number of S.P.C : 594

LOAD Computation

Name: Loads.1

Applied load resultant :

Fx	=	1	.	702e-015	N
Fy	=	9	.	810e+002	N
Fz	=	-5	.	956e-015	N
Mx	=	9	.	258e-008	Nxm
My	=	-4	.	353e-016	Nxm
Mz	=	-1	.	027e-008	Nxm

STIFFNESS Computation

Number of lines	:	37890			
Number of coefficients	:	1321830			
Number of blocks	:	3			
Maximum number of coefficients per bloc	:	499962			
Total matrix size	:	15	.	27	Mb

SINGULARITY Computation

Restraint: Restraints.1

Number of local singularities	:	0
Number of singularities in translation	:	0
Number of singularities in rotation	:	0
Generated constraint type	:	MPC

CONSTRAINT Computation

Restraint: Restraints.1

Number of constraints	:	594
Number of coefficients	:	0
Number of factorized constraints	:	594
Number of coefficients	:	0
Number of deferred constraints	:	0

FACTORIZED Computation

Method	:			SPARSE
Number of factorized degrees	:	37296		
Number of supernodes	:	2123		
Number of overhead indices	:	215565		
Number of coefficients	:	7759440		
Maximum front width	:	828		
Maximum front size	:	343206		
Size of the factorized matrix (Mb)	:	59	.	1998
Number of blocks	:	8		
Number of Mflops for factorization	:	2	.	919e+003
Number of Mflops for solve	:	3	.	122e+001
Minimum relative pivot	:	4	.	154e-004

Minimum and maximum pivot

Value	Dof	Node	x (mm)	y (mm)	z (mm)
5.6070e+006	Tx	12630	2.5100e-005	5.5412e-006	2.0317e+002
3.7319e+010	Tx	5484	7.4853e+000	-6.4531e+001	-3.4531e+002

Minimum pivot

Value	Dof	Node	x (mm)	y (mm)	z (mm)
9.3669e+006	Tx	12626	4.8100e+000	5.4199e+001	-4.4597e+002
3.4214e+007	Tx	6432	2.4470e+000	-5.4017e+001	3.6078e+002
4.7467e+007	Tx	12629	4.8100e+000	3.7952e+001	-4.4549e+002
5.5111e+007	Tx	12423	9.5192e-006	1.3701e+001	-5.4715e+001
5.5813e+007	Ty	7790	1.8455e+001	7.2398e+001	-2.0970e+002
6.2778e+007	Ty	7026	-1.5585e-005	1.3701e+001	4.7671e+002
6.5049e+007	Ty	1332	-4.8100e+000	2.7402e+001	2.6568e+002
8.1350e+007	Tx	11279	5.3087e+000	-7.1445e+001	1.2963e+002
8.3074e+007	Tx	4489	-4.8100e+000	4.7134e+001	-3.8289e+002

Translational pivot distribution

Value	Percentage
10.E6--> 10.E7	5.3625e-003
10.E7--> 10.E8	3.7538e-002
10.E8--> 10.E9	5.8827e+000
10.E9 --> 10.E10	9.1967e+001
10.E10--> 10.E11	2.1075e+000

DIRECT METHOD Computation

Name: Static Case Solution.1
Restraint: Restraints.1
Load: Loads.1
Strain Energy : 9.738e-004 J

Equilibrium

Components	*Applied Forces*	*Reactions*	*Residual*	*Relative Magnitude Error*
Fx (N)	*1.7018e-015*	*9.7233e-011*	*9.7235e-011*	*1.9226e-012*
Fy (N)	*9.8100e+002*	*-9.8100e+002*	*3.2333e-010*	*6.3931e-012*
Fz (N)	*-5.9562e-015*	*4.5247e-011*	*4.5241e-011*	*8.9456e-013*
Mx (Nxm)	*9.2576e-008*	*-9.2537e-008*	*3.8975e-011*	*1.5413e-012*
My (Nxm)	*-4.3527e-016*	*-8.6928e-012*	*-8.6933e-012*	*3.4378e-013*
Mz (Nxm)	*-1.0275e-008*	*1.0278e-008*	*3.1703e-012*	*1.2537e-013*

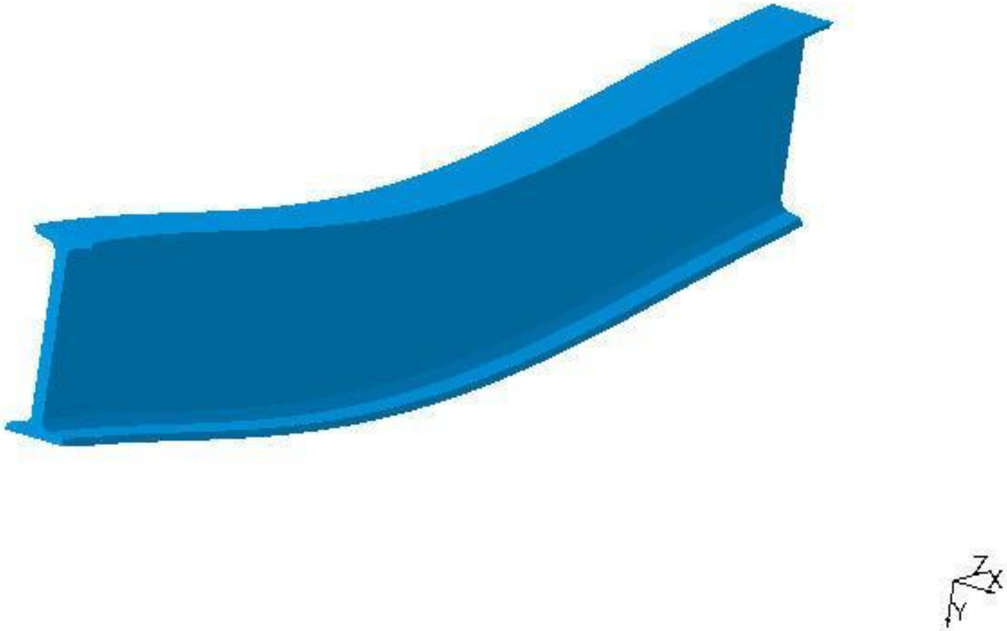

Figure 2

On deformed mesh ---- On boundary ---- Over all the model

Static Case Solution.1 - Von Mises stress (nodal values).2

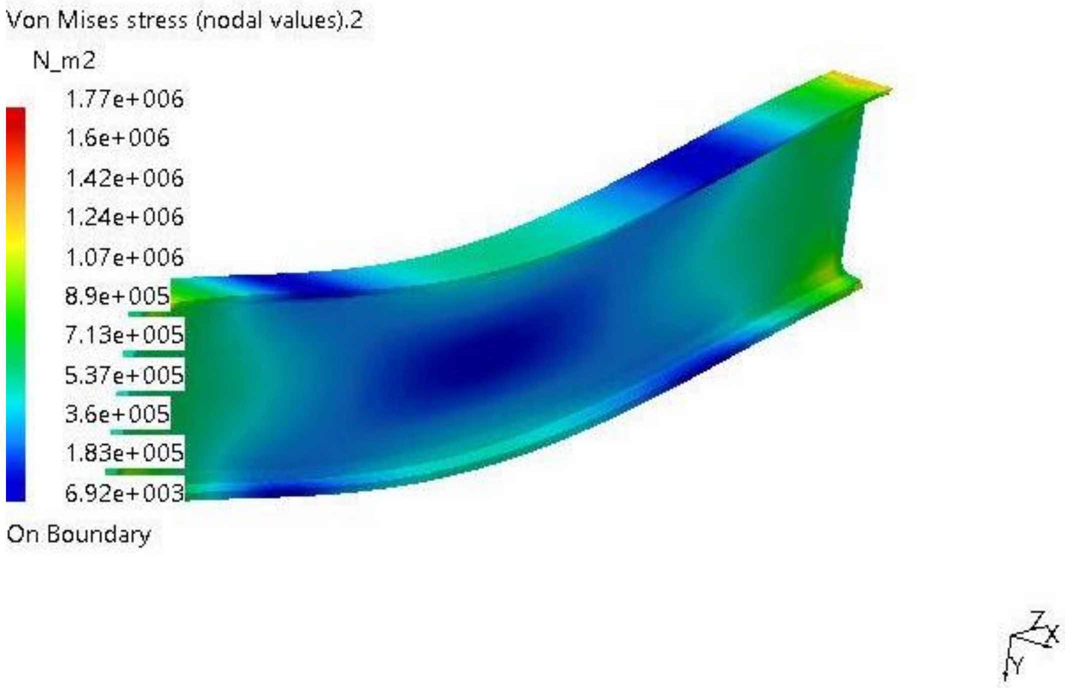

Figure 3
3D elements: : Components: : All
On deformed mesh ---- On boundary ---- Over all the model

Global Sensors

Sensor Name	Sensor Value
Energy	9.738e-004J