Structural Specialist Design Challenge Solution

Written by Godswill Ezeorah,

As part of the JobApplication Process

At SpaceRyde.

Questions Provided by David Platt (Structures team lead)

Q1 Load Factor Design Approach

Consider the following assumptions:-

- → Negligible motor gimble force
- → Ignoring members selfweight
- → Joint technique not fully expressed
- → Thrust force in horizontal x-direction.

As shown below the free body diagram of the Primary load bearing Structure;

$\omega = 43.3^\circ$ (taken at the tangent of the curved beam towards its end points)

~~Lg = 50 in~~

lac = 20 in

~~$L_e = \sqrt{50^2 + 20^2}$ in~~

~~lac = 20 in = Lc~~

~~$\phi = \tan^{-1}\left(\frac{2 \times la}{Lg}\right)$~~

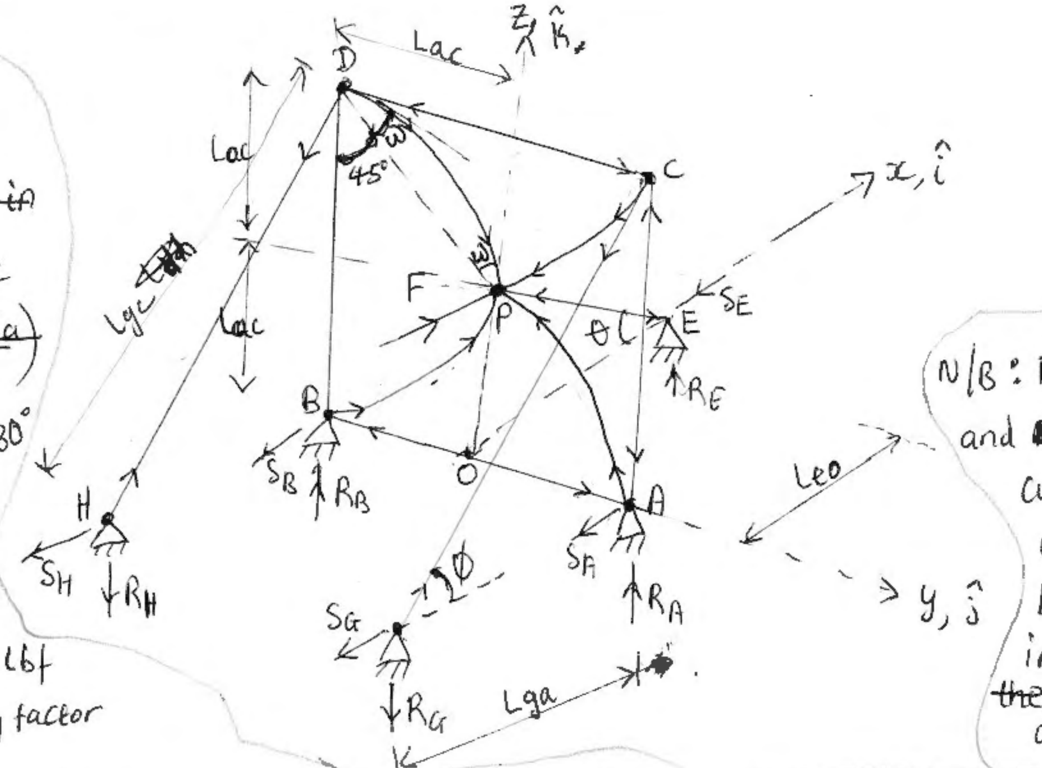
$\theta = 20^\circ$; $\phi = 30^\circ$

~~$L_{eo} = \sqrt{50^2 - 20^2}$ in~~

$L_{ga} = \frac{2 l_{ac}}{\tan(\phi)}$

$F = 20{,}000 \times 3$ lbf
safty factor

$L_{eo} = \frac{2 l_{ac}}{\tan(\theta)}$



N/B: members ⇒ F_{PA}, F_{PC}, F_{PB}, and F_{PD} are all curved beam, which are a bit tricky, in terms of the design approach

⇒ Joint force analysis method:-

Moment about the origin-O

$M_{io} \Rightarrow R_A = R_B$ and $R_G = R_H$

Hence members are symmentric about the x-z plane.

$M_{jo} \Rightarrow F \times La = R_E \times L_e + 2 R_G \times L_g$ ⟵

Nomenclature:

R_A: Reaction force at joint A in z-direction

S_A: Reaction force at Joint A in x-direction

F_{BA}: Internal force acting on member BA.

F : Thrust force

Q1 cont'd.)

@ Joint P

$\Sigma_{iP} \Rightarrow F = 2F_{PA} + 2F_{PC} + F_{PE}$ ← // ** considering the symmetric nature ** //

$\Sigma_{KP} \Rightarrow F_{PA} - F_{PC} + \frac{F_{PE}\sin\theta}{2\times(\cos\omega)(\cos 45)}$ ←

@ Joint C

$\Sigma_{iC} \Rightarrow F_{PC}(\sin\omega) = F_{GC}(\cos\phi)$ ←

$\Sigma_{jC} \Rightarrow F_{PC}(\cos\omega)(\cos 45) = F_{DC}$ ←

$\Sigma_{KC} \Rightarrow F_{CA} - F_{GC}(\sin\phi) - F_{PC}((\cos\omega)(\cos 45) = 0$ ←

@ Joint E

$\Sigma_{KE} \Rightarrow R_E = F_{PE}(\sin\theta)$ ←

$\Sigma_{iE} \Rightarrow S_E = F_{PE}(\cos\theta)$ ←

@ Joint G

$\Sigma_{Kg} \Rightarrow R_G = F_{GC}(\sin\phi)$ ←

$\Sigma_{ig} \Rightarrow S_G = F_{GC}(\cos\phi)$ ←

@ Joint A

$\Sigma_{iA} \Rightarrow F_{PA}(\sin\omega) = S_A$ ←

$\Sigma_{jA} \Rightarrow F_{BA} = F_{PA}((\cos\omega)(\cos 45)$ ←

$\Sigma_{KA} \Rightarrow R_A = F_{CA} - F_{PA}(\cos\omega)(\cos 45)$ ←

Q1) cont'd.)

Solving the above eqns with the ← symbol, using MATLAB symbolic function as shown in Appendix-A, will yield the following member forces:-

Member	Force (Lbf)	Dominate Structural Behaviour
BA	2030.611	Compressive
CA, DB	11630.570	Compressive and bending
DC	6573.070	Compressive
GC, HC	10115.001	Tensile
PA, PB	3945.898	Share and bending
PC, PD	12772.839	Share and bending
PE	26562.525	Compressive

As well as the corresponding reaction forces at the member support:-

Notation	Support	Reaction Forces (Lbf)	Direction
R	A, B	9599.960	Vertical -z
R	E	9084.919	Vertical -z
R	G, H	5057.500	Vertical -z
S	A, B	2706.169	Horizontal -x
S	E	24960.609	Horizontal -x
S	G, H	8759.848	Horizontal -x

The above result is verified by taking the sum of moment about any point (in this case, Joint-A). And it resulted in zero.

∴ The equilibrium condition is satisfied.

Q2 cont'd

* Members under bending load

Such as CA, DA

We solve for the sectional elastic modulus,

$Z = \frac{M}{\sigma_y}$, with M as the bending moment expirienced by the member (e.g $M_{ca} = 2l_a \times F_{pa}$

This has been solved using the code in Appendix-A and the resulting sectional modulus are:

$Z_{ca} = Z_{da} = 62.348\ cm^3$ (x-x axis)

using this for selection also results in the section of

IPE 140 A

* Curved beam case

Members PA, PB, PC and PD, profile selection was done by chosing a customs solid profile. Whose section area ~~parameter~~ perimeter are same to it's equivalent elastic sectional modulus result of;

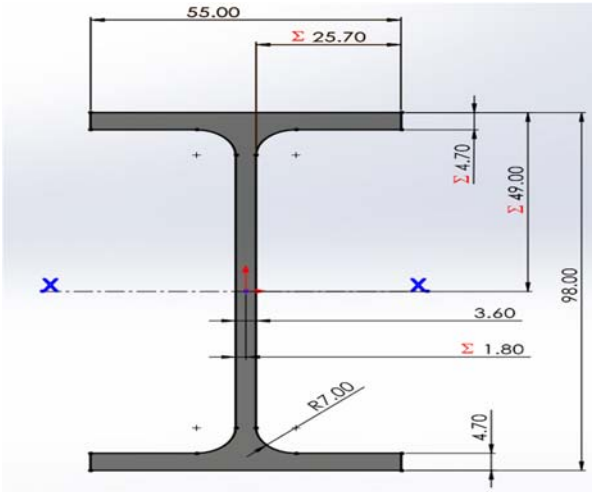
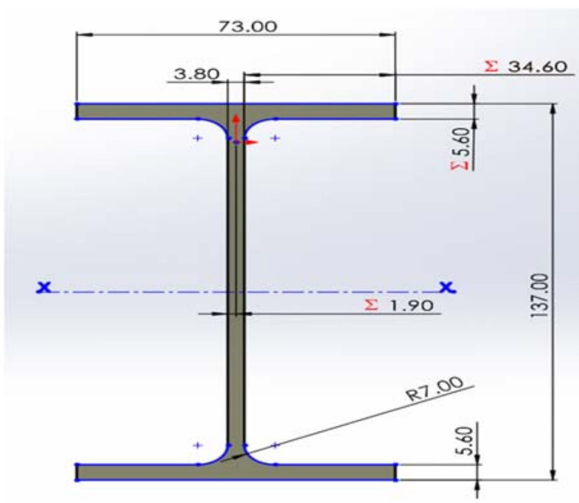
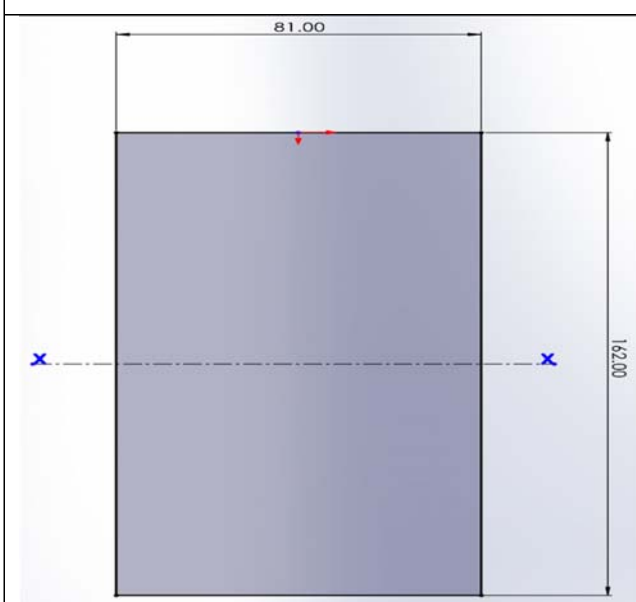
$Z_{pa} = 34.595$

$Z_{pc} = 111.975\ cm^3$ ← (x-x axis)

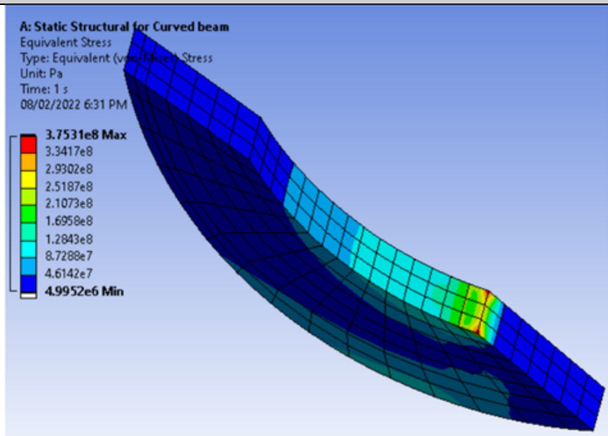
~~$Z_{ca} = 62.345$~~

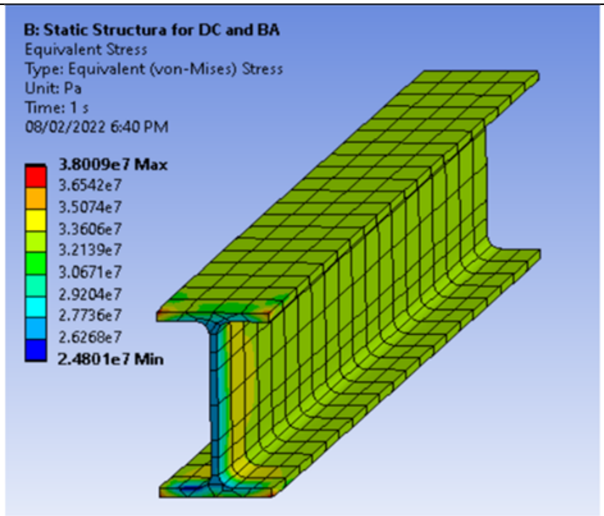
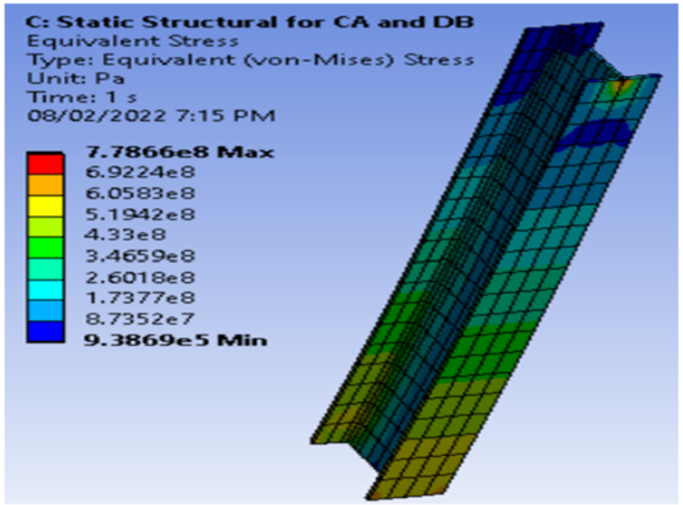
which leads to the selection of IPE 160R, but due to the greater effect of shearing at the joint, a solid profile is selected.

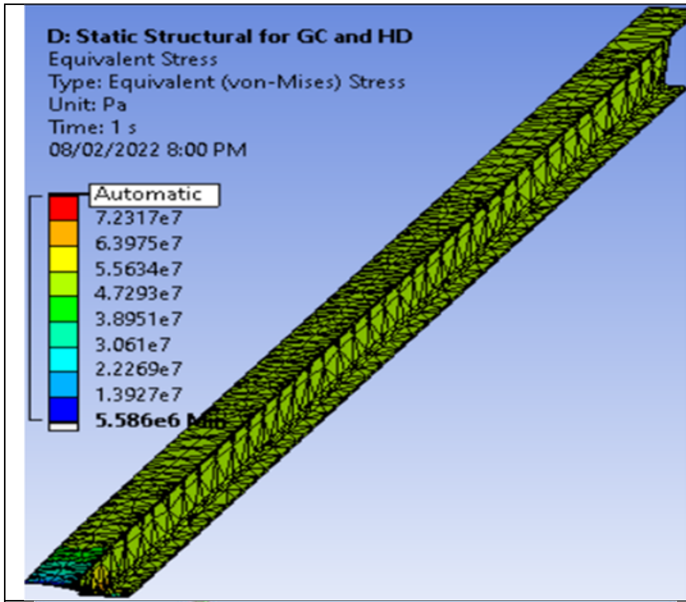
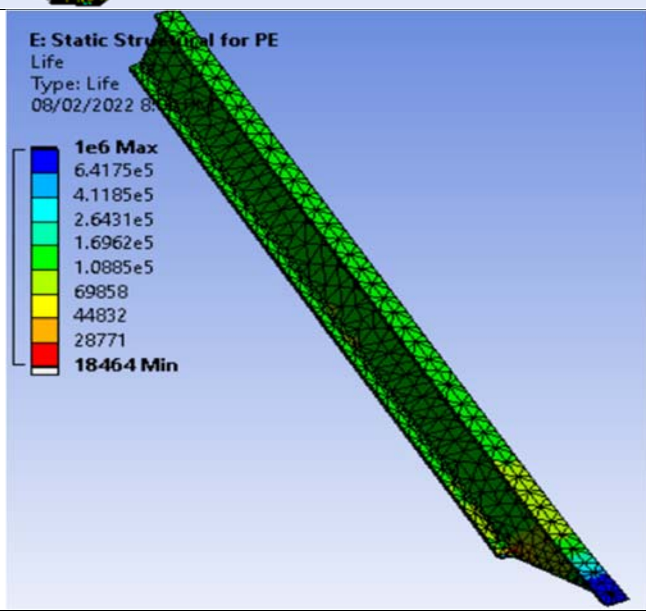
N/B: To improve the structural performance of these curved member, special care must be given to it joint technique

Section Profile	European code	Member
	IPE 100A	BA, DC, GC, HC, and PE
	IPE 140 A	CA and DA
	Custom profile (from IPE 160 R Perimeter)	PA, PB, PC and PD

Finite element method is used to analyze the structural failure of each member, using Ansys software. The results are described in the table below:

Result	Member	Maximum Equivalent Stress (mPa)	Average Equivalent Stress (mPa)	Minimum Life Cycle, N	Remark
	PA, PB, PC and PD	375.31	42.468	3292.5	The low cycle life is as a result of the sharing damage closer to the support region

	BA and DA	38.009	33.167	1.e+006	Sound structure, since the maximum von misses stress is less that the steel material compressive yield strength
	CA and DB	778.66	264.72	455.98	The member acts as interface between three other members, hence its failure is due to bulking caused by other members internal forces. (This member is the weak point the entire structure)

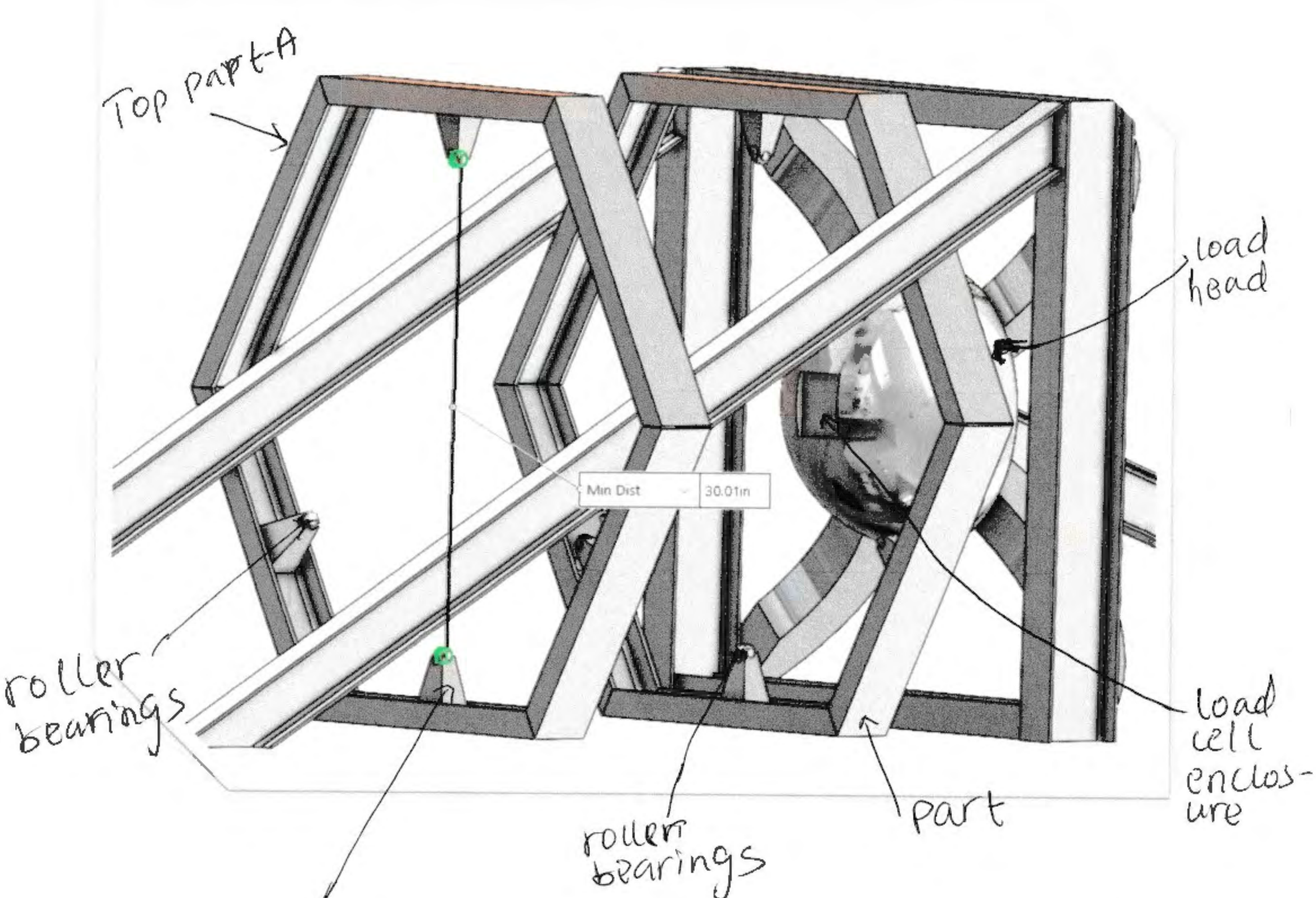
	GC and HD	80.658	50.576	1.e+006	Sound structure
	PE	219.05	132.87	18464	Sound structure, and yet it bears approximately 1/3 of the thrust load

Q4| The motor is mounted by the following steps :-

=> open top part-A

=> using folk lift/machinary, drop the motor on bottom part-A.

=> slide into part-B, and ~~unto~~ forward onto the Load cell enclosure.

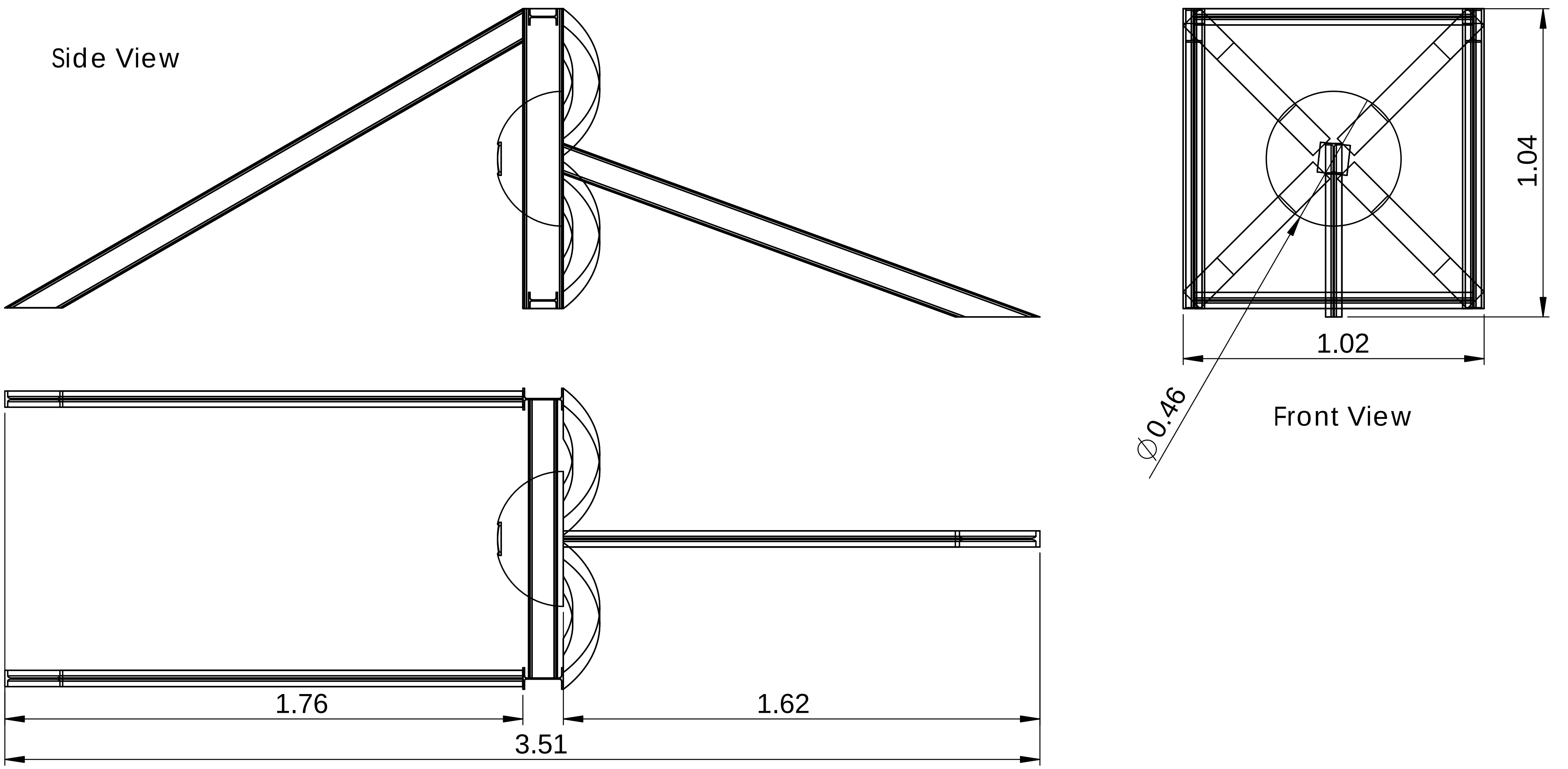


In the case of damage of the expensive ~~roller~~ bearing, spring/damping part can be used inplace of this part.

The following consideration was taken into account when designing the members:

- Ability to withstand the safety factored thrust load, with very minimal tradeoff to cost.
- The use of standardized beam members will save time during design implementation and certification from regulatory bodies.
- Compact design dimensions (as shown on page 12)

Thus, to justify the member selection, the assembled frame structure is analyzed as a whole using the finite element method. The result shows that the average equivalent stress is 51.453 mPa which is less than the material yield strength, Hence the member selection is justifiable. And the resulting deformation simulation is shown in the video below:



UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:	FINISH:	DEBURR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING	REVISION		
			UNIT: m			
	NAME	SIGNATURE	DATE		TITLE:	
DRAWN	G.E.	G.E.	02/08/22		ROCKET TEST STAND DESIGN - HORIZONTAL	
CHK'D						
APPV'D						
MFG						
Q.A				MATERIAL: Structural Steel (S185)	DWG NO. 01	A4
				WEIGHT: 23.37 lbs	SCALE:1:20	SHEET 1 OF 1

Q 5

Since the deformation occur in the elastic regime, the stress life approach of fatigue analysis is used in Ansys software to compute the life of the assembled ~~stru~~ test stand structure. With the applied Factored load of about 60,000 lbf, the resulting fatigue life is averaged as 856,890 cycles, and a minimum cycle of about 1678 localized at members CA and DB, which is as a result of the larger structural damage in those members.

∴ Yes the design will maintain the safety factor for 100 hot fires and for 1000 hot fires. But will fail at 1678 hot fires (if ~~one~~ 1 hot fire = 1 fatigue cycle, N)

APPENDIX - A

```
%**Code written by Godswill on 09 Feb 2022, for solve test**%
%**stand problem for horizotal oriented recket motor**%




clear; clc;
format long g;
% unit conversion factors
Funit=4.44822161526;  %from Lbs to N
Dunit=2.54; % from inches to cm


% Initialization of Variables
F=60000; theta=20; phi=30; w=43.3;
La=20; %unit in inches
L=Dunit*sqrt(40^2+40^2)/2; %unit in cm
Leo=La/tand(theta); %unit in inches
Lga=2*La/tand(phi); %unit in inches
Lgc=sqrt(Lga^2+(2*La)^2); %unit in inches
Lpe=sqrt(Leo^2+(2*La)^2); %unit in inches


syms Re Ra Rg Sa Se Sg Fpc Fgc Fpa Fpe Fdc Fba Fca
mv=F*20==Re*Leo+2*Rg*Lga;
ip=F==2*Fpa+2*Fpc+Fpe;
kp=Fpa-Fpc+(Fpe*sind(theta)/(2*cosd(w)*cosd(45)))==0;
ic=Fpc*sind(w)==Fgc*cosd(phi);
jc=Fpc*cosd(w)*cosd(45)==Fdc;
kc=Fca-Fgc*sind(phi)-Fpc*cosd(w)*cosd(45)==0;
ke=Re==Fpe*sind(theta);
ie=Se==Fpe*cosd(theta);
kg=Rg==Fgc*sind(phi);
ig=Sg==Fgc*cosd(phi);
ia=Fpa*sind(w)==Sa;
ja=Fba==Fpa*cosd(w)*cosd(45);
ka=-Fpa*cosd(w)*cosd(45)+Fca==Ra;


sol=solve(mv,ip,kp,ic,kc,ie,ke,ig,kg,ia,ka,jc,ja);


digits(5)
Fca=double(sol.Fca)
Fgc=double(sol.Fgc)
Fpa=double(sol.Fpa)
Fpc=double(sol.Fpc)
Fpe=double(sol.Fpe)
Fdc=double(sol.Fdc)
Fba=double(sol.Fba)
Ra=double(sol.Ra)
Re=double(sol.Re)
```

```
Rg=double(sol.Rg)
Sa=double(sol.Sa)
Se=double(sol.Se)
Sg=double(sol.Sg)


% Taking Moment @ joint A for verification
verify_sol=round(F*20-Re*Leo-2*Rg*Lga)


T=table({'Re';'Ra';'Rg';'Sa';'Se';'Sg';'Fpc';'Fgc';'Fpa';'Fpe';'Fdc';'Fba';'Fca'},
...
    [Re; Ra; Rg; Sa; Se; Sg; Fpc; Fgc; Fpa; Fpe; Fdc; Fba; Fca]);
filename = 'result.xlsx';
writetable(T,filename,'Sheet',1,'Range','A1');


%** Section selection depending on dorminate stress (axial or bending)**%%
% structural steel mechanical properties
seg_y=25000; %in N/cm^2
seg_t=25000; %in N/cm^2






% Members with axial load as control for section selection
Ape=Fpe*Funit/seg_y  %compression
Agc=Fgc*Funit/seg_y  %tensile
Adc=Fdc*Funit/seg_t  %compression
Aba=Fba*Funit/seg_t  %compression
% Members with bending load as control for section selection
Mpc=L*Fpc*sind(w)*Funit;
Zpc=round(Mpc/seg_y,3)
Mpa=L*Fpa*sind(w)*Funit;
Zpa=round(Mpa/seg_y,3)
Mca=2*La*Fpc*sind(w)*Funit;
Zca=round(Mca/seg_y,3)


% Converting forces to Newton
Fpc_N=round(Fpc*Funit*sind(w),3) % unit in N
Fpe_N=round(Fpe*Funit,3) % unit in N
Fdc_N=round(Fdc*Funit,3) % unit in N
Fgc_N=round(Fgc*Funit,3) % unit in N
```