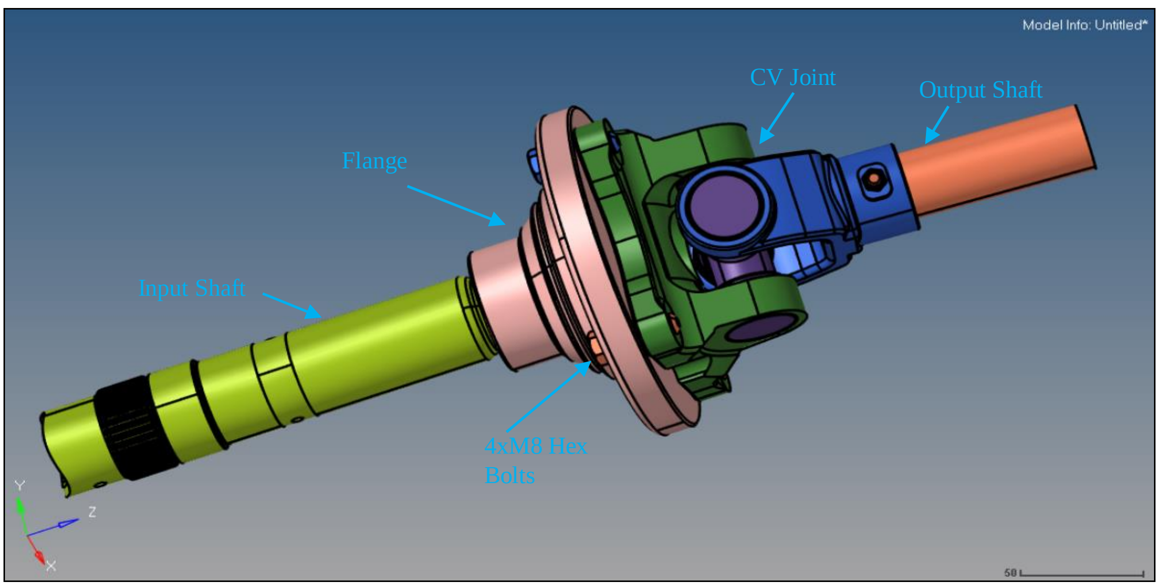
# Mass Reduction & Structural Optimization of the Automobile Transmission System

#FEA #HyperMesh #Abaqus #Automobile #Powertrain

- ***Description:***

This project aims to conduct the CAE stress analysis and topological optimization on the transmission sub-system of the powertrain system of the front-wheel drive cars with the aid of HyperMesh pre-processing and Abaqus solver. As the picture shown, the transmission subsystem is simplified to 1:1 scale with some main features, input/output steel-tube drive shafts, the flange, the CV Joint and its mating parts and bolts. In this project, the CV Joint *(Constant Velocity Joint)* is designed to transmit a 2500Nm torque to the wheel end with a 10$^{o}$ offset angle between input and output shafts. The meshes are well and smoothly generated with 50,000+ nodes and fine control size-wise and layer-wise, especially at the concentrated load areas, ie. fillet, spline teeth. For better approximation to the real working condition, 2 load steps, ie. Bolt Pre-tension and Torque are applied. Given the system is regarded as rigid body, the torque is applied at the equivalent point of CV Joint with the constraints of CONN3D2 connector and COUP_KIN rigid coupling. To make sure the analysis can yield a good convergence, the system is restrained with the Soft Spring method based on Hook's Law. As what the result indicates that the stress is concentrated at the contact area of the bolts and the optimized structure is yielded with 28% profile cut off from the original hub design.

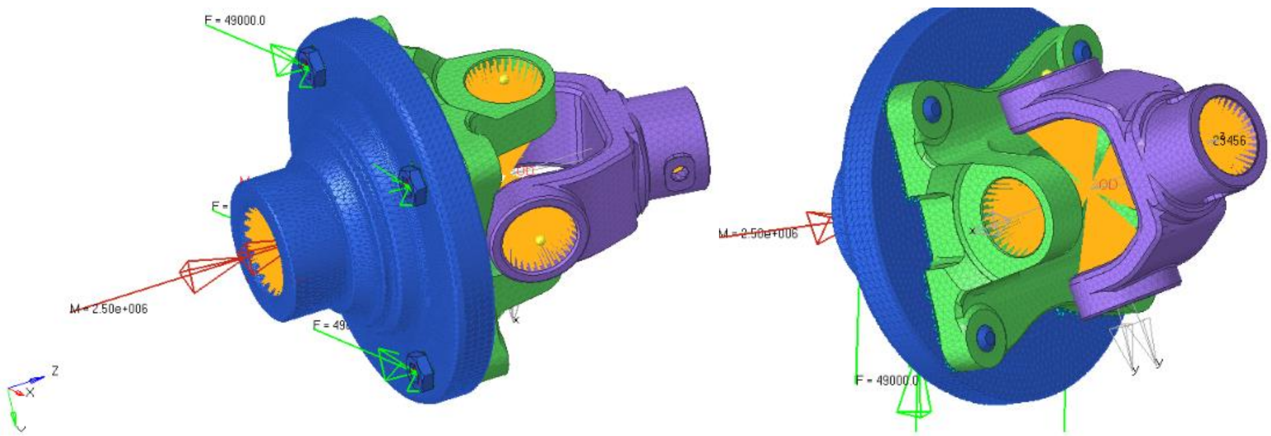



- ***Meshes** (Components & Loads)*

The .stp geometric models are imported into HyperMesh to create 2D and 3D (Tetra) meshes. As shown in the picture below, the stress concentration of fillet areas are captured with 3 layers with 1mm R-trias (right angle) to make up the grid distortion. Because the mechanical properties of the pretension load, the bolts are captured with the Hex grid. The input shaft is neglected and its force is transmitted into the flange hence the flange is constrained with the COUP_KIN rigid coupling to withstand the movement and the 2500Nm torque of CONN3D2 connector. Therefore, the movement constraints of the mating part are also set by the 49kN pre-tension load setting of 4xM8 bolts with the flange. And two mating parts are coupled on equivalent point to transmit torque to the output shaft. The other components are meshed with size-5 regular trias for the speed of computation. According to Hooke's Law, F=Kx, the large-stiffness spring will offset the movement when the FEA models are hard to converge. The steel material is edited with 210 GPa, 0.28 poisson ratio under solidsection model image and the contact surfaces are tied together with 0.15 surface friction and are checked with 0.1 positon tolerance to see how the meshes interfere with each other.

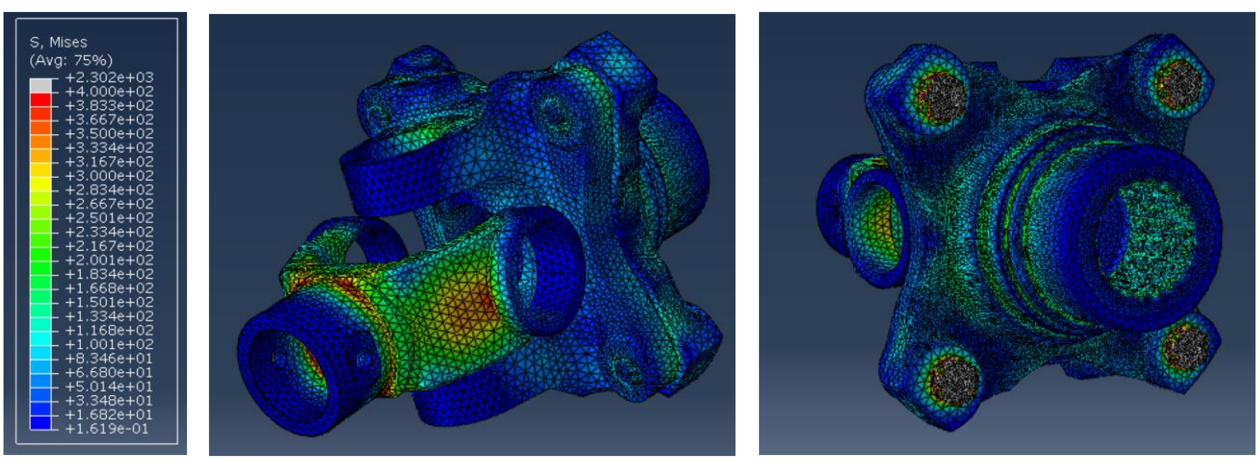
**Components Meshes**


**Load Meshes & Constraints**




- ***Result***

After importing to Abaqus Job manger, the structure is optimized to the structure with only bolt areas remaining. In this way, the mass reduction yields to be 28% of previous model without any load greater than the material's yield strength. The Von-Mises stress of the model is shown below.



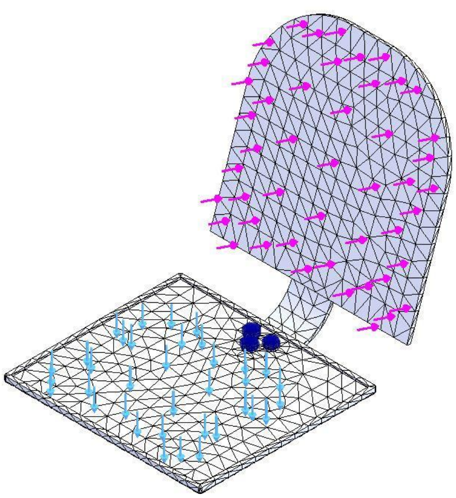
# Design and Static Stress Analysis of General-Purpose Office Chair

- ***Purpose:***

To design the model of a fixed seat-angle type office chair and analyze the mechanical capabilities under impact test and sustainably requirement conforming to ANSI/BIFMA X5.1-2011 code.

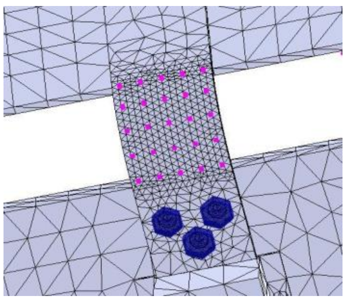
- ***Approach:***

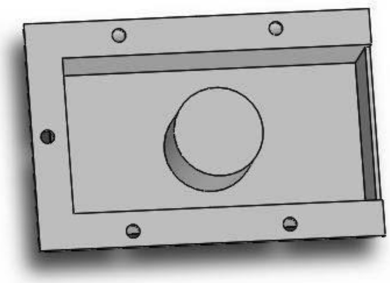
1. The general assembly of the office chair is separated in 3 parts for analysis, seat & bracket, armrest and seating star base.
2. In terms of the design, alloy steel is selected due to its elastic Modulus, and high tensile strength, Balsa wood is selected for the seat for the combination of low cost and comfort. design of five-le, ABS is selected due to good impact resistance and the ease of injection molding. The gged star seating base is adopted for adequate stable support and are inclined to tipping. Thus, Al-4032 five-legged star base is better choice with stability. ABS is selected due to good impact resistance and the ease of injection molding.



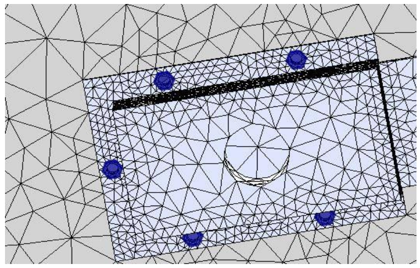
3. In terms of Simulation, the seat & base is tested with normal sitting condition with a 600N force applied downwards to the seat and 300N normal to the bracket. The connection area is set as fixed geometry for the constraints of the simulation. And for a more reliable result with a fast processing time, Mesh Control is used to set the finer mesh at the bracket of backrest, which endures more bending load as a sheet-metal piece and at the lip of the connection part where the self-tapping screws sit, endures more stress so it needs finer mesh to capture the stress profile.

**Mesh Control**

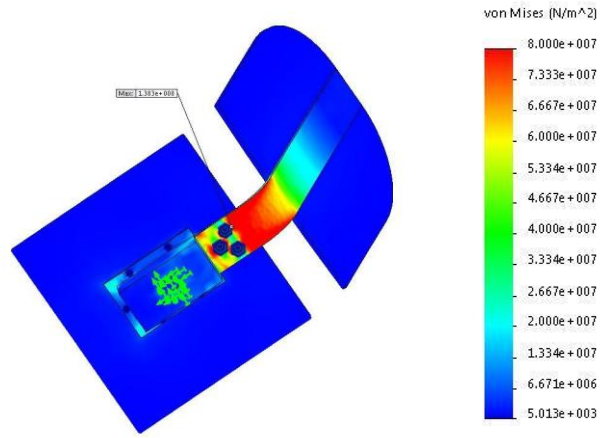


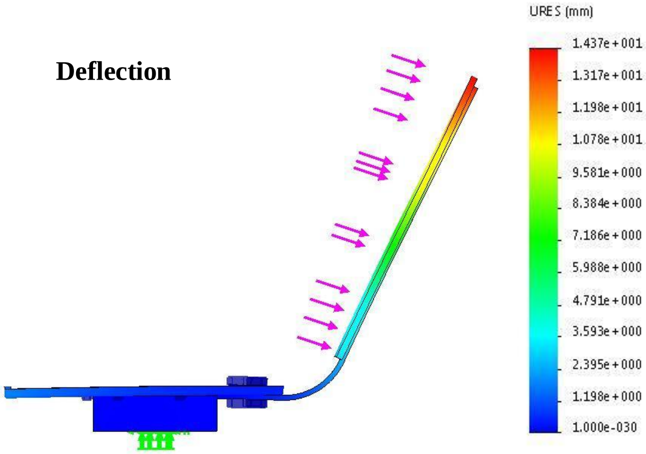


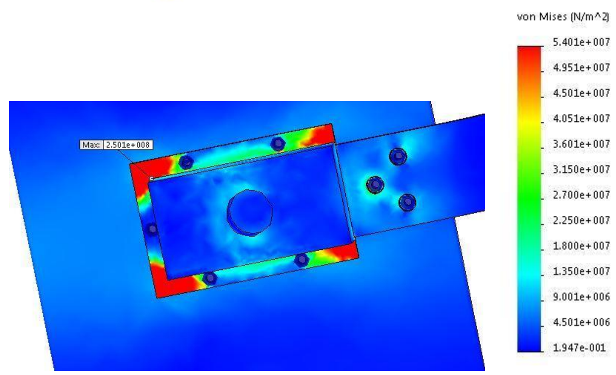
**Mesh Control**



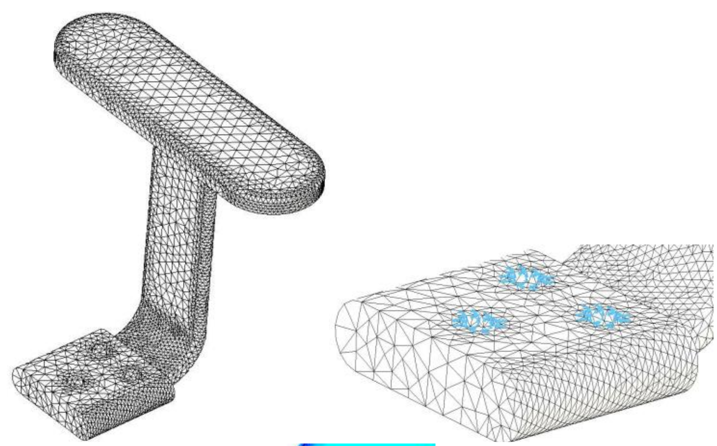
4. Von Mises stress and deflection is applied to check if the maximum bending stress reach the yield strength and to show bending deflections. As shown in the results, the max. stress occurs at the cantilevered end of the bracket with 138 MPa and max. deflection is at the top of backrest with 1.48 mm. The max. stress exerted on the connection lip is 250 MPa which leads to Safety of Factor to be 1.67 given 415MPa yield strength of Alloy Steel, Al-4140.

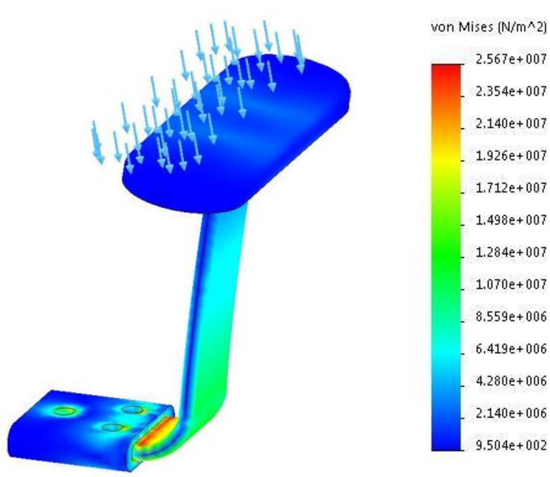


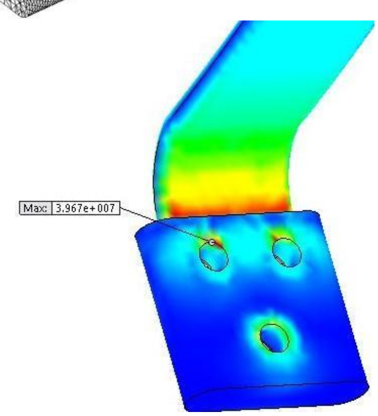



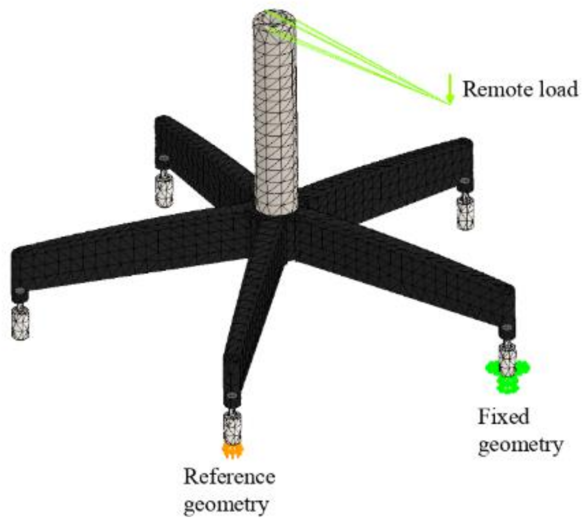

5. The test of the armrest is applied with a 400N repetitive load to the top with 10-degree angle from inward to simulate the worst bending condition caused by human arms. As the result shown, the max. stress of the arm is exerted at the cantilevered end of the arm with 25.6 MPa and on the bolt holes with 39.6 MPa.



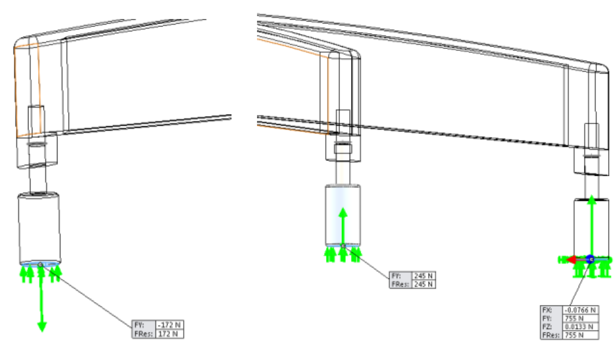


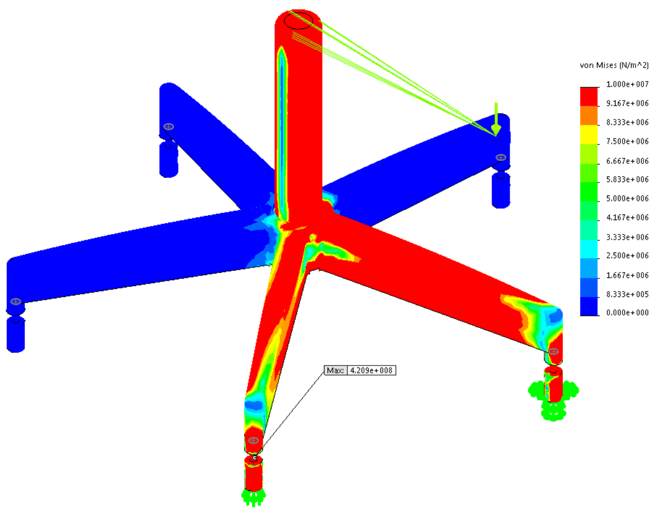
6. An extreme test is applied to simulate the tilting condition of the star base with 1000N remote load downward at the edge of the seat, ie. 0.3m from the center of the shaft. It is assumed that one wheel is defined as fixed geometry, meaning cannot move with respect to ground. And then select other wheels as reference geometries which can move horizontally to the ground. After simulation, the function of listing result force is used to check the assumption. As shown in result, there is one force value occurs to be minus. It is impractical that the ground can only provide the supporting force with upwards direction. Then, this leg is eliminated from the reference geometries list. After checking the iteration, the final tilting condition turns to be 1 fixed geometry, 1 referenced geometry and other legs are lifted floating. From results, the fixed wheel reacts more supporting force than moved one. Then, stress and displacement simulations are carried out.



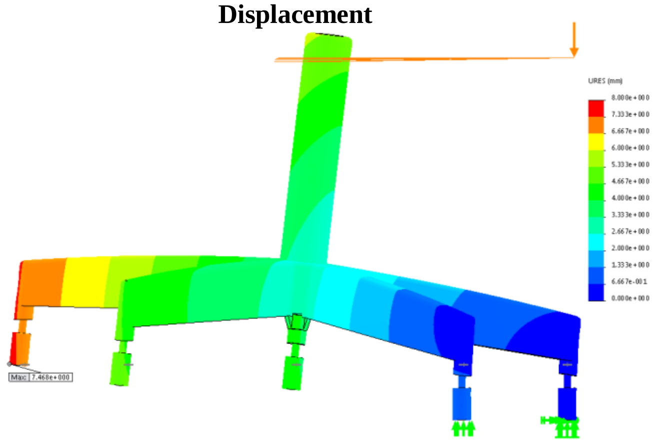

**Check Reaction Forces to Verify Assumption**



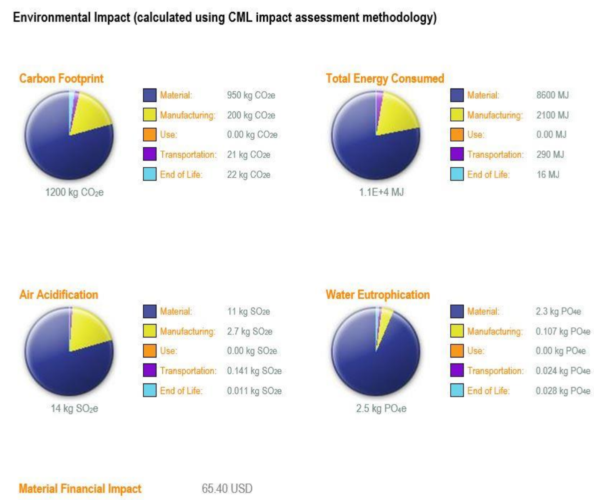
**Von Mises Stress**



**Displacement**



**Sustainability Analysis**

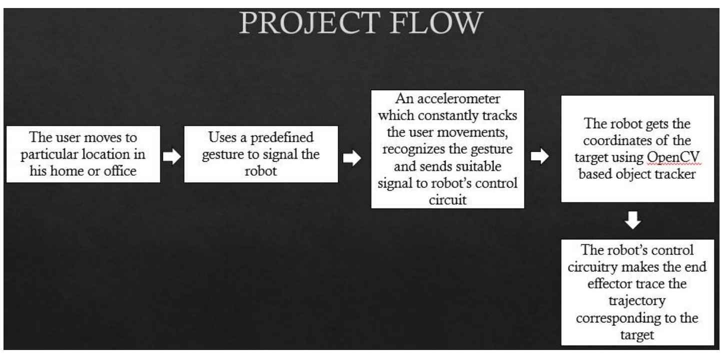


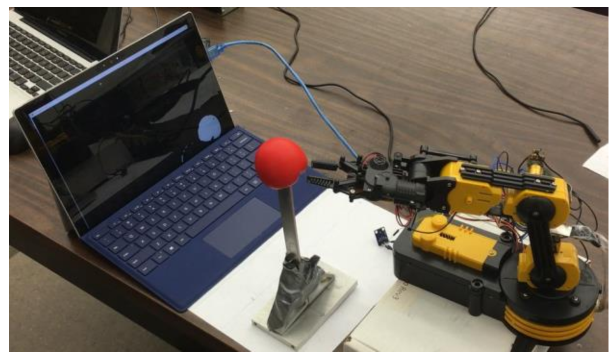
# A Robotic Prototype as Prosthesis with Control & Computer Vision

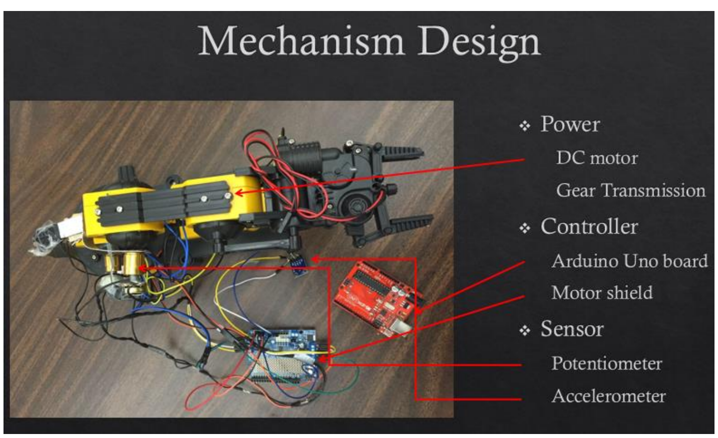
#Robotics #Arduino #Kinematics #Matlab #OpenCV

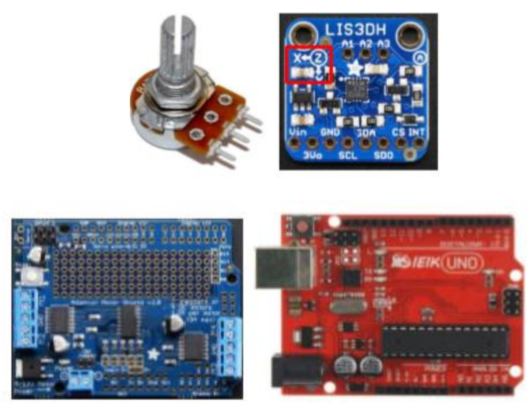
- ***Description:***

This project aims to design and control a color-oriented and force-triggered 4-DOF FANUC robot to work as a prosthesis to grab objects for patients using computer vision. It contains 3 basic topics, Forward/Inverse Kinematics, Trajectory planning and PID control. Our team assembled a simple 4-DOF robot (OWI Robotic Arm) which is driven by steering engine and mounted on accelerometer for trigger and potentiometer for joints data reading using Arduino microcontroller. The forward and inverse kinematic of its workspace is simulated in Matlab and then a trajectory of the goal of Red object is planned to make the arm perform a specific motion by using OpenCV to filter the object out of the surroundings.





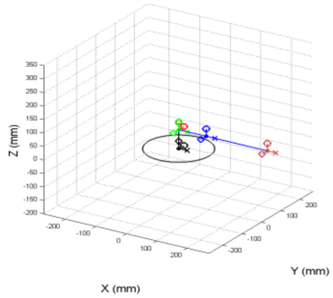



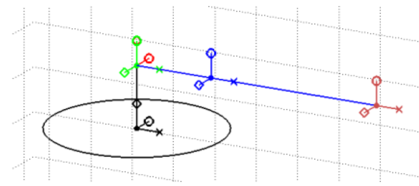

- ***Forward Kinematics & Inverse Kinematics***

The 4 DOF robot is represented with DH parameters and put into Matlab. The configuration of (0,0,0,0) joint angles are shown below. Then one arbitrary Cartesian point, (100, 150, 200) is given to the matlab simulation composed of equations below to calculate back the joint spaces reversely to be ($0^o$, $56^o$, $100^o$, $-88^o$). The matlab simulation works correctly and then the code is deployed into the Arduino microcontroller and Adafruit motor shield. After initialization and predefine the macro parameters, the accelerometer_sense() reads the jerk from the accelerometer which simulates the patient can stamp to trigger the action with. Then the system using OpenCV HSV values to define the coordinate of the end goal. After the arm reaches the target and checking with potentiometer reading of joint values, the clamp() will execute to grab the target.

### Forward Kinematics

| DH parameters | | | | |
|---|---|---|---|---|
| **i** | $a_{i-1}$ | $\alpha_{i-1}$ | $d_i$ | $\theta_i$ |
| **1** | 0 | 0 | L1 | $\theta_1$ |
| **2** | 0 | $90^\circ$ | 0 | $\theta_2$ |
| **3** | L2 | 0 | 0 | $\theta_3$ |
| **4** | L3 | 0 | 0 | $\theta_4$ |



### Inverse Kinematics

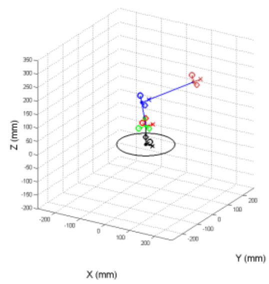
$$\theta_1 = \mathrm{atan2}(p_y, p_x)$$

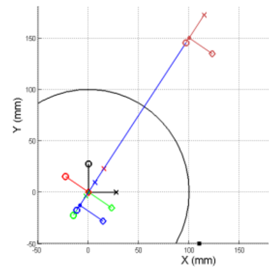
$$\cos\theta_3 = \frac{(p_x{}^2 + p_y{}^2 + (p_z - L1)^2 - (L2^2 + L3^2))}{(2 \cdot L2 \cdot L3)}$$

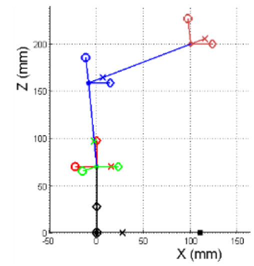
$$\theta_3 = \mathrm{atan2}(\pm\sqrt{1 - \cos\theta_3{}^2}, \cos\theta_3)$$

$$\cos\alpha = \frac{(p_x{}^2 + p_y{}^2 + (p_z - L1)^2 + L2^2 - L3^2)}{(2 \cdot L2 \cdot \sqrt{(p_x{}^2 + p_y{}^2 + (p_z - L1)^2})} \quad \text{offset angle}$$

$$\theta_2 = \mathrm{atan2}\left((p_z - L1), \sqrt{p_x{}^2 + p_y{}^2}\right) + \mathrm{atan2}(\pm\sqrt{1 - \alpha^2}, \cos\alpha)$$




- ***Results and Code***

1. Trajectory --- https://www.youtube.com/watch?v=SFZeGK11iKA

2. Accelerometer --- https://www.youtube.com/watch?v=OxU7ojJStO8

3. Visualization --- https://www.youtube.com/watch?v=aaaq6QB4oI4

In conclusion, The robot can generally function to satisfy the design purpose. But, the data noise has an effect which will impede the speed and accuracy of the motion. It is lucky the noise isn't accumulated too much to offset the expected target coordinate. The filter like low-pass or Kalman Filter (LQE) are better choices for more accurate data acquisition and control.

# Design and control of A Robotic Manipulator

This project aims to design a robotic manipulator with different length combinations of multibody system composed of flexible body and rigid body for dynamics simulation and analysis. And using the piezo-ceramic sheet as a smart transducer to feedback the bending vibration signals as sensor and adjusting to accurate position functioning with linear amplifier as actuator. A 2 dof manupulator with detachable is designed in CREO and its natural frequency, amperage caused by bending stress are calculated. Then the hardware is established and programed with LabVIEW SoftMotion module to actuate and PID control.

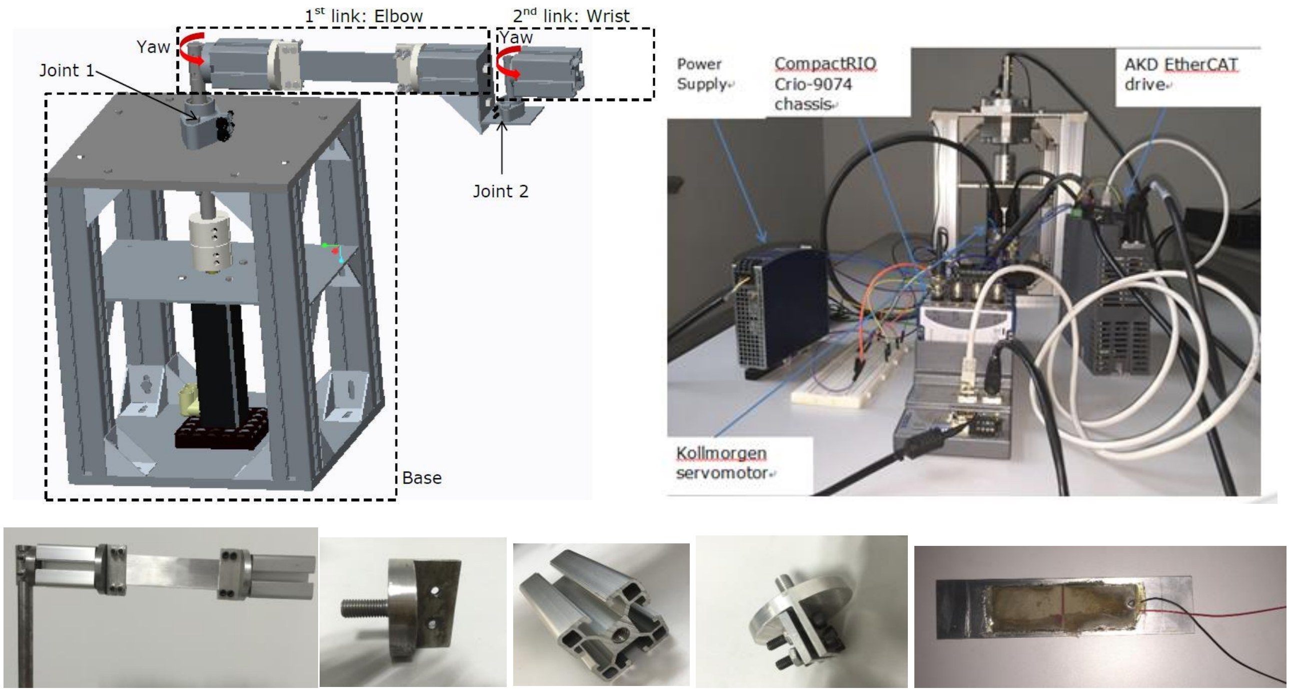



Figure 1. Components

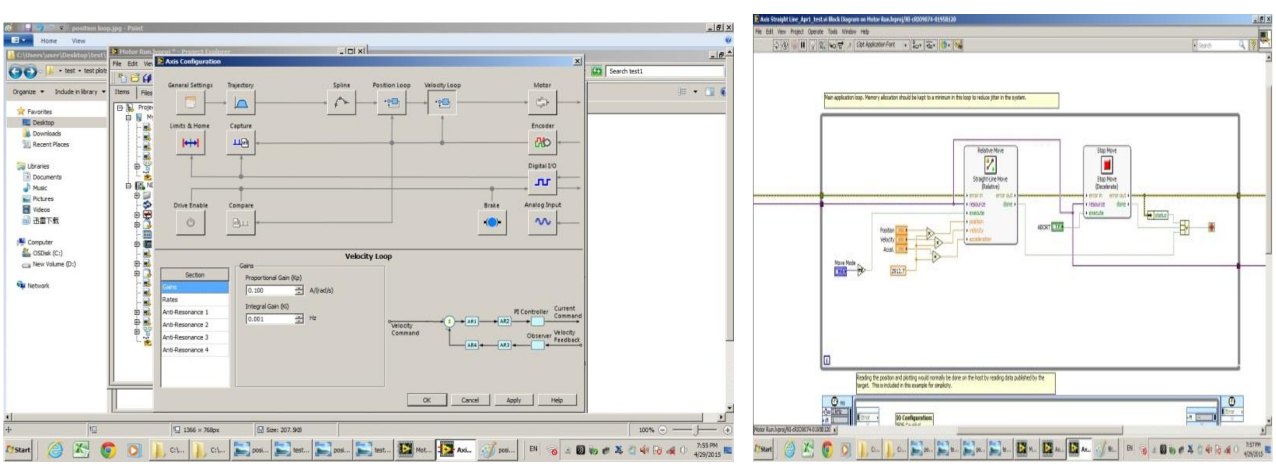
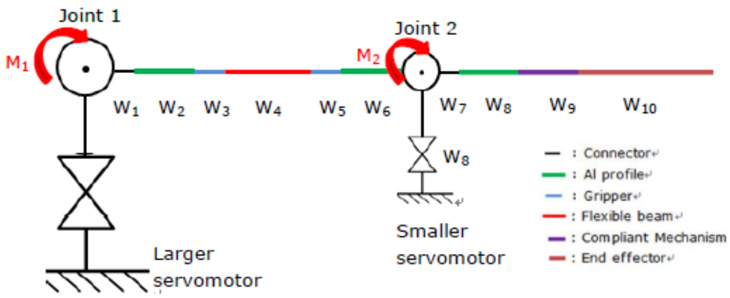


Figure 2. LabView Softmotion Module




$$\mathbf{M_1} = (W_1 \times L_1 + W_2 \times L_2 + W_3 \times L_3 + W_4 \times L_4 + W_5 \times L_5 + W_6 \times L_6 + W_7 \times L_7 + W_8 \times L_8 + W_9 \times L_8 + W_{10} \times L_{10}) \cdot g$$

The torque equation about joint 2:

$$\mathbf{M_2} = (W_7 \times L_7{}' + W_8 \times L_8{}' + W_9 \times L_9{}' + W_{10} \times L_{10}{}') \cdot g$$

**Frequency equation for particular end conditions**

| | |
|---|---|
| Pinned-pinned | $\sin \lambda L = 0$ |
| Clamped-clamped & free-free | $\cos \lambda L \cosh \lambda L - 1 = 0$ |
| Clamped-pinned & free-pinned | $\tan \lambda L - \tanh \lambda L = 0$ |
| Clamped-free | $\cos \lambda L \cosh \lambda L + 1 = 0$ |

**Numerical values of roots, $\lambda_r L$, of frequency equations**

| $r$ | 1 | 2 | 3 | 4 | 5 | >5 |
|---|---|---|---|---|---|---|
| Pinned-pinned | $\pi$ | $2\pi$ | $3\pi$ | $4\pi$ | $5\pi$ | $r\pi$ |
| Clamped-clamped & free-free* | 4.730 | 7.853 | 10.996 | 14.137 | 17.279 | $\approx (r + 0.5)\pi$ |
| Clamped-pinned & free-pinned | 3.927 | 7.069 | 10.210 | 13.351 | 16.493 | $\approx (r + 0.25)\pi$ |
| Clamped-free | 1.875 | 4.694 | 7.855 | 10.996 | 14.137 | $\approx (r - 0.5)\pi$ |

* A free-free beam will also have 2 rigid body modes corresponding to $\lambda L = 0$.

Selecting the values of $\lambda_r L$ from the above table for the beam of interest, the natural frequencies can be found from equation (5). That is: $\omega_r = \frac{(\lambda_r L)^2}{L^2} \sqrt{\frac{EI}{\rho A}}$

a) Estimated natural frequency for 1st link

| | $\omega_{flexible}$ (Hz) | $\omega_{whole}$ (Hz) |
|---|---|---|
| 1st mode | 246.85 | 74.38 |
| 2nd mode | 1547.12 | 466.18 |
| 3rd mode | 4332.42 | 1305.46 |

Table 1. The results of the natural frequency of the 1st link

b) Estimated natural frequency for two-link arm

| | $\omega_{flexible}$ (Hz) | $\omega_{whole}$ (Hz) |
|---|---|---|
| 1st mode | 246.85 | 34.80 |
| 2nd mode | 1547.12 | 218.11 |
| 3rd mode | 4332.42 | 610.78 |

Table 2. The results of the natural frequency of the 2nd link

Figure 3. Calculation of Natural Frequency and Bending Moments

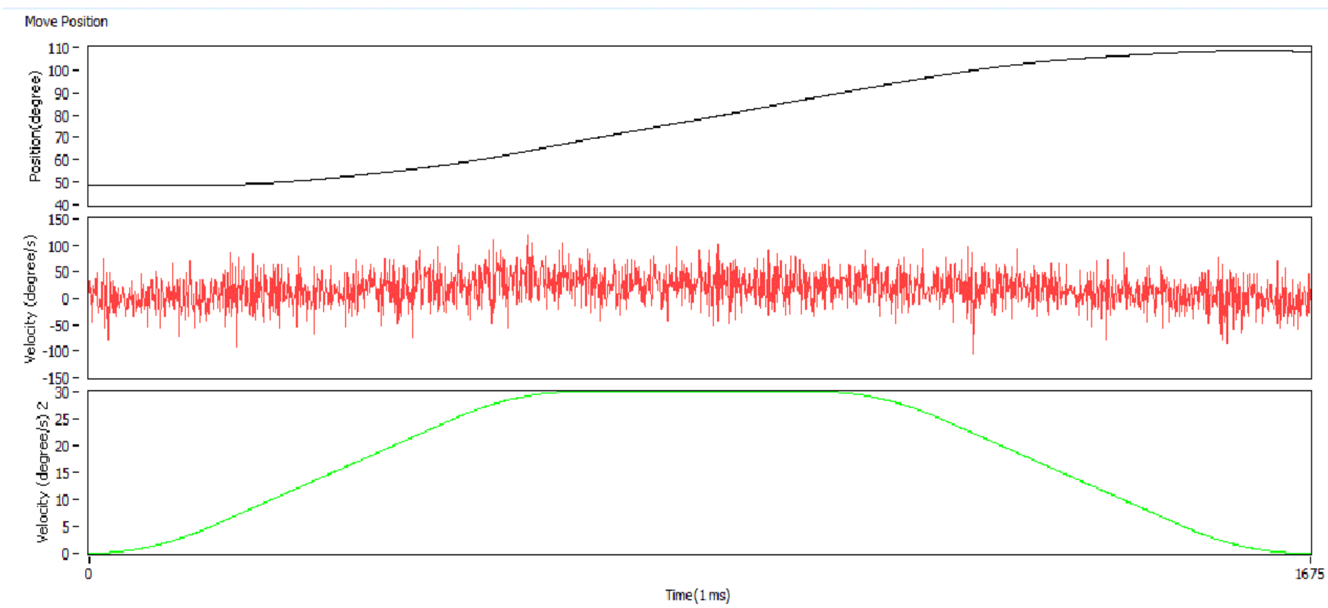
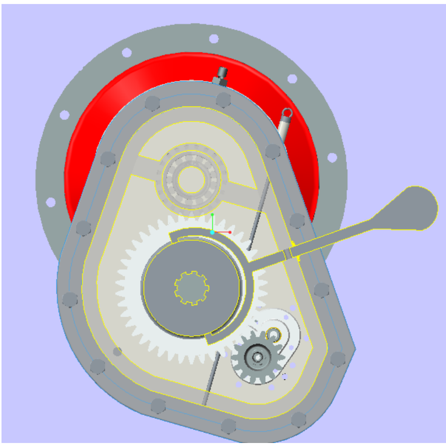


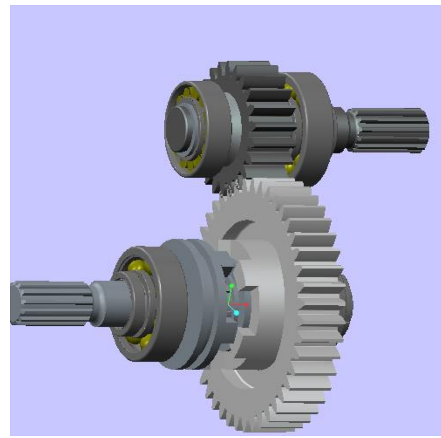

Figure 3. Data Acquistion from the Servo Motor

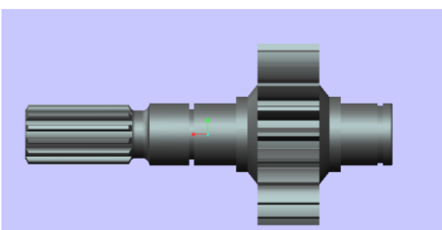
# ***Other Works***

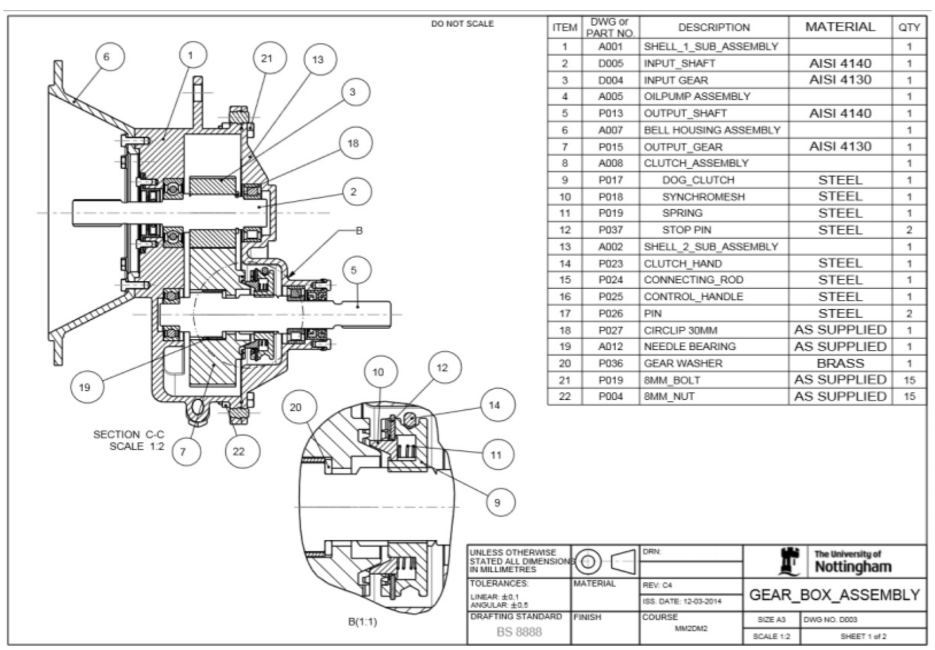
## Marine Gearbox Design

This project aims to design a gearbox suitable for a small displacement pleasure craft and inshore fishing boat. The design intent of the gearbox is 20.8kw @ 3600rpm/ 1700rm max torque, lightweight and high tightness. The calculations of gear module, the contact stress are correctly performed. This design engages the axial bearing, lip seal, dowel pins, dog clutch to disengage the transmission, oil pump and oil detection stick.



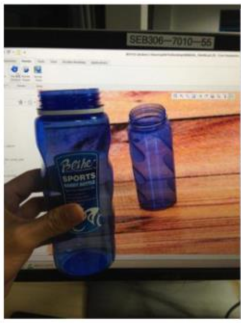


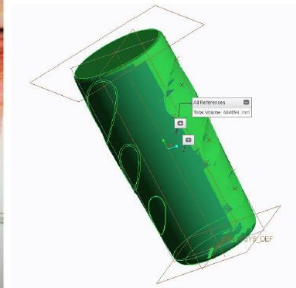





| ITEM | DWG or PART NO. | DESCRIPTION | MATERIAL | QTY |
|---|---|---|---|---|
| 1 | A001 | SHELL_1_SUB_ASSEMBLY | | 1 |
| 2 | D005 | INPUT_SHAFT | AISI 4140 | 1 |
| 3 | D004 | INPUT GEAR | AISI 4130 | 1 |
| 4 | A005 | OILPUMP ASSEMBLY | | 1 |
| 5 | P013 | OUTPUT_SHAFT | AISI 4140 | 1 |
| 6 | A007 | BELL HOUSING ASSEMBLY | | 1 |
| 7 | P015 | OUTPUT_GEAR | AISI 4130 | 1 |
| 8 | A008 | CLUTCH_ASSEMBLY | | 1 |
| 9 | P017 | DOG_CLUTCH | STEEL | 1 |
| 10 | P018 | SYNCHROMESH | STEEL | 1 |
| 11 | P019 | SPRING | STEEL | 1 |
| 12 | P037 | STOP PIN | STEEL | 2 |
| 13 | A002 | SHELL_2_SUB_ASSEMBLY | | 1 |
| 14 | P023 | CLUTCH_HAND | STEEL | 1 |
| 15 | P024 | CONNECTING_ROD | STEEL | 1 |
| 16 | P025 | CONTROL_HANDLE | STEEL | 1 |
| 17 | P026 | PIN | STEEL | 2 |
| 18 | P027 | CIRCLIP 30MM | AS SUPPLIED | 1 |
| 19 | A012 | NEEDLE BEARING | AS SUPPLIED | 1 |
| 20 | P036 | GEAR WASHER | BRASS | 1 |
| 21 | P019 | 8MM_BOLT | AS SUPPLIED | 15 |
| 22 | P004 | 8MM_NUT | AS SUPPLIED | 15 |

## CAD Rendering of Bottles





## Escalator Handrail Switch Inlet Cover