International Journal of Research in Advent Technology, Vol.2, No.2, February 2014

E-ISSN: 2321-9637

Estimation of Stresses at the Crucial Parts of a Coupling

Dr.G.D.Mehta[1], Vir Bahadur[2]

Mechanical Department[1, 2],Priyadarshini College of Engineering, Nagpur[1, 2]

Email:girishm97@rediffmail.com[1]

Abstract-In this paper stress analysis of a flexible coupling is presented. The flexible coupling is used in a cutter machine which is generally used for cutting aluminum bloom. The cutter machine is facing the problem of coupling failure during its operation. Hence, a Finite Element approach is used to ascertain the main reason behind the failure of coupling.

Index Terms- Finite element analysis; flexible coupling; analysis through Ansys software.

1. BACKGROUND OF PRESENT WORK

The present paper basically emphasizes on the case pertaining to a cutter machine. The operation of cutter machine is explained as under. When one wants to cut an aluminium bloom into different pieces, he places the aluminium bloom to a moving bed and then he pushes a button to operate a machine. The cutter machine consists of a cutter unit, which is coupled with a motor shaft through a flexible coupling. Due to rotational movement of a cutter an aluminium bloom is then cut. The process of cutting is done continuously for one shift and if in any case some break down take place during the production process, then it may result in production loss.

Presently, this breakdown is occurring in industries frequently. Thus, it is required to find out the reason for this phenomenon.

Logical reasons for the above stated problem can be:

(1) The power of the motor is in the tune of 50 HP, which creates high driving torque.

(2) Aluminium bloom surface provides process resistance to the cutter, which in turn produces a load torque at the cutter shaft. This load torque is then transferred to the motor shaft.

(3) A coupling becomes a node point, where the coupling experiences load torque. To overcome this load torque, motor produces additional driving torque. Here, at this point coupling again experiences the increase in driving torque. This process is continued during the process of cutting.

(4) Due to fluctuation of load, coupling gets failed.

Hence in this regards, estimation of deflections and its corresponding stresses are done in the following way:

(1) The quarter portion of the coupling is considered for conventional F.E.A. approach, where the total quarter portion is converted into two dimensional sections. This section is then discritised into nine different finite elements. By applying constraints and different forces at respective nodes, the global stiffness and force matrix are obtained. These matrices are useful for finding out the

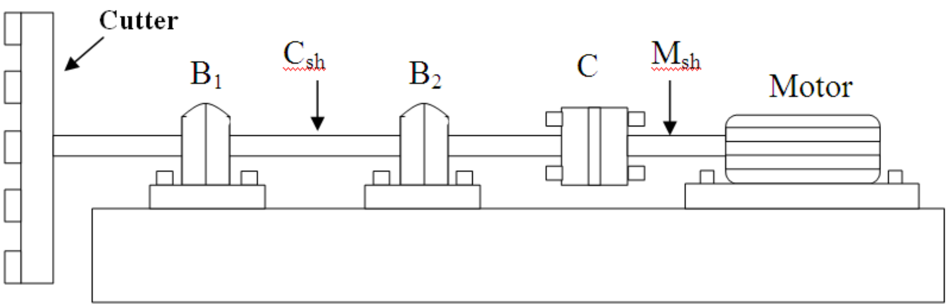
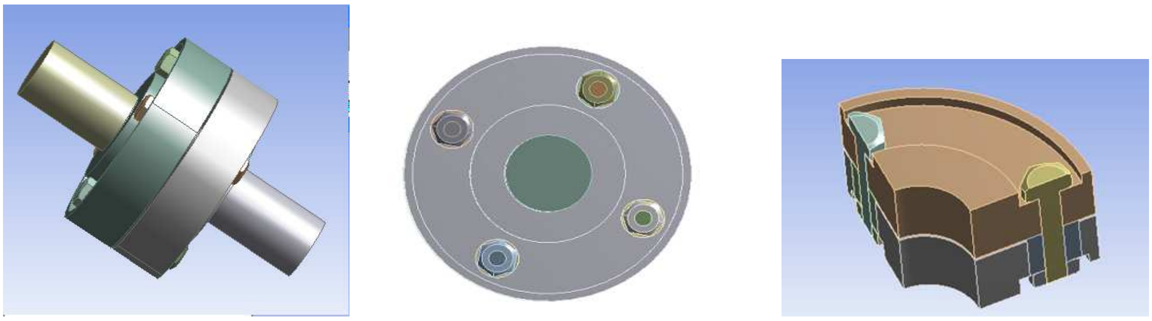


Fig. 1. Schematic arrangement of cutter machine



a. Pictorial view   b. Front view   c. Quarter section

Fig. 2. Flexible coupling.

deflections and stresses at different nodal position.

(2) In ANSYS approach, the total quarter portion of a coupling is discritised in 3D elements. Applying the constraints & forces at respective nodes, deflections & stresses are then estimated.

(3) Lastly, the obtained result by a FEA approach is compared with ANSYS Approach.

2. MAJOR ESTMATION OF DEFLECTION AND STRESSES OF FLEXIBLE COUPLING: FEA APPROACH

2.1. *Generation of 2 D (two dimensional) section of a flexible coupling*

Figure 2(a) shows a pictorial view of a flexible coupling. If one is looking from a front direction of a coupling, then a front view can be obtained as shown in Figure 2(b). It is now required to generate 2-D section of a coupling for FEA approach. The section of a coupling is circular, which is shown in Figure 2(c).

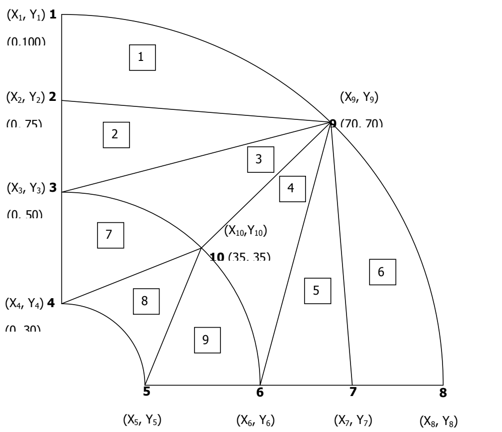


Fig.3. Representation of discritized two dimensional elements

Hence, one can consider a quarter section of Figure 2(a) which is shown in Figure 2(c). Now the quarter cutting section is discritized in to small finite elements.

2.2. *Discritization of 2D section*

In the present work the quarter portion of flexible coupling is discritized into 9 finite elements (Refer Figure5).

Every element is numbered by a proper sequence. An example for one such element No. 1 is given in figure 6.

There are three local nodes as well as three global nodes as shown in Figure5. The complete list of local and global nodes and their corresponding distances from reference point (0, 0) are given in Table1.

Table 1. Element connectivity table.

Elements	1	2	3	Local node
1	1	2	3	Global node
2	2	3	9	
3	3	10	9	
4	10	6	9	
5	6	7	9	
6	7	8	9	
7	3	4	10	
8	4	5	10	
9	5	6	10	

2.3. *Application of constraints*

Now, it is essential to apply the constraints to a cut portion of a flexible coupling. Due to driving torque the force F1 is produced at node 2 which is nothing but a force given by a bolt to a flange body.

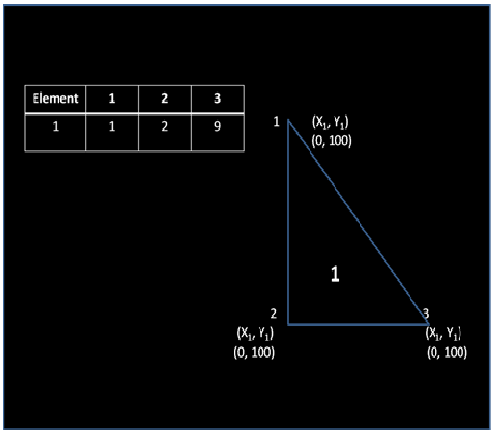


Figure6: Systematic representation of element number 1 with its local and global nodes

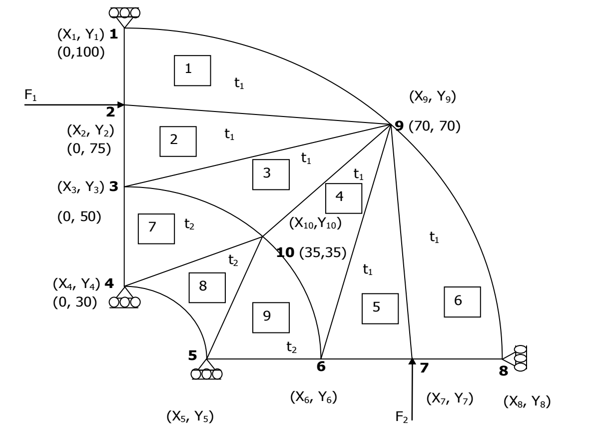


Figure7: Application of constraints to the quarter portion of a flexible coupling

While force F2 is the reaction force produced by bolt on a flange body, which is applied at node 7. (In the present case F1=F2=824 N, which is calculated on torque basis).

At node 1 and 8 roller supports are provided so that the deflection in X direction at node 1 will be accountable, while in Y direction the deflection at node 8 will be accountable. Similarly at node 4 and 5 roller supports are provided so that deflection in X direction will be accountable.

2.4. *Formulation of material property matrix [D], Jacobian matrix [j], strain displacement matrix [B] & element stiffness matrix [K] Sub-headings*

The next step is to formulate Material Property Matrix [D], Jacobian Matrix [J], Strain Displacement Matrix [B] & Element Stiffness Matrix [K]for nine elements. In view of page restriction, one sample calculation for element 1 is given belowSub-headings.

Formulation of Material Property Matrix [D]:

$$D_1 = \frac{E}{1-\mu^2}\begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix}$$

In the above Matrix:

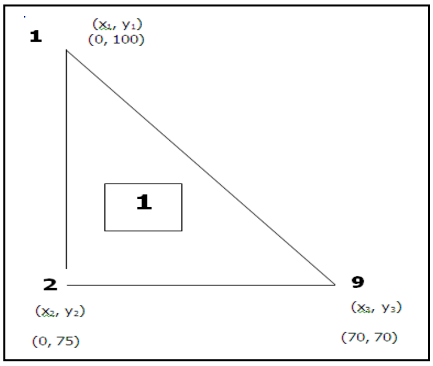
E = Young's Modulus of Elasticity

μ = Poisson's ratio

The value of E for (SAE 1030) Material is 200 x 10^3 N/mm^2 and value of μ is 0.3.

After substituting the above values [D] matrix is converted into:

$$D_1 = \begin{bmatrix} 219780 & 65934 & 0 \\ 65934 & 219780 & 0 \\ 0 & 0 & 76923 \end{bmatrix}$$



Formulation of Jacobian Matrix [J]:

$$|J_1| = \begin{bmatrix} X_{13} & Y_{13} \\ X_{23} & Y_{23} \end{bmatrix}$$

$$|J_1| = \begin{bmatrix} -70 & 30 \\ -70 & 5 \end{bmatrix}$$

$$|J_1| = 1750 \text{ mm}^2$$

$$A_1 = |J_1|/2 = 875 \text{ mm}^2$$

Formulation of Strain Displacement Matrix [B]:

$$B_1 = 1/J_1 \begin{pmatrix} Y_{23} & 0 & Y_{31} & 0 & Y_{12} & 0 \\ 0 & X_{32} & 0 & X_{13} & 0 & X_{21} \\ X_{32} & Y_{23} & X_{13} & Y_{31} & X_{21} & Y_{12} \end{pmatrix}$$

$$B_1 = \begin{pmatrix} 0.0029 & 0 & -0.0171 & 0 & 0.0143 & 0 \\ 0 & 0.0400 & 0 & -0.0400 & 0 & 0 \\ 0.0400 & 0.0029 & -0.0400 & -0.0171 & 0 & 0.0143 \end{pmatrix}$$

$$B^T_1 = \begin{pmatrix} 0.0029 & 0 & 0.0400 \\ 0 & 0.0400 & 0.0029 \\ -0.0171 & 0 & 0.0400 \\ 0 & -0.0400 & 0.0171 \\ 0.0143 & 0 & 0 \end{pmatrix}$$

Formulation of Element Stiffness Matrix $[K]_e$
$[K]_e = t.A.[B]^T[D]^T[B]$ .In the above equation
t = thickness of plate element;A = Area of the element;$[B]^T$ = Transpose of [B] Matrix;$[D]^T$ = transpose of [D] Matrix

$$K^1 = 1 \times 10^7 \begin{pmatrix} 0.4370 & 0.0571 & -0.4684 & -0.2110 & 0.0314 & 0.1538 \\ 0.0571 & 1.2330 & -0.1890 & -1.2440 & 0.1319 & 0.0110 \\ -0.4684 & -0.1890 & 0.6568 & 0.3429 & -0.1884 & -0.1538 \\ -0.2110 & -1.2440 & 0.3429 & 1.3099 & -0.1319 & -0.0659 \\ 0.0314 & 0.1319 & -0.1884 & -0.1319 & 0.1570 & 0 \\ 0.1538 & 0.0110 & -0.1538 & -0.0659 & 0 & 0.0549 \end{pmatrix}$$

2.5. *Estimation of deflection*

After formulating an element stiffness matrix for each element, the next step is to assemble the global stiffness matrix. The global matrices represent the complete body or continuum. This global stiffness matrix is of size [20 X 20]. After applying boundary conditions, the global stiffness matrix is reduced from its original size [20 X 20] to [16 X 16] .Then, by using reduced global stiffness matrix and force matrix, one can determine the unknown displacements by using following equation

[a]=[b] x [c]

Where [a] - force matrix

[b] – reduced global stiffness matrix

[c] – Nodal displacement matrix

In the present case, reduced global stiffness matrix[a] is of size [16 x 16]. While, force matrix [a] is of [16 x1]

These matrices are then solved by using Mat-lab software and thus following nodal displacements are obtained (values of displacements are in mm).

1X 0.0001775; 2X 0.0001734; 2Y 0.0000103; 3X 0.0000491

3Y 0.0000097; 4X -0.0000592; 5X -0.0000523; 6X -0.0000645

6Y 0.00006567; X -0.0000272; 7Y 0.0001530; 8Y 0.0001339

9X 0.0000403; 9Y 0.00000981; 0X -0.00002861; 0Y 0.0000037

3. ESTIMATION OF DEFLECTION AND STRESSES OF FLEXIBLE COUPLING: ANSYS APPROACH.

3.1. *Construction of 3-d object in pro-e software Numbering and spacing*

Basically Pro-Engineers Software is used for the generation of 3 Dimensional image of complex engineering components. The software has several specialty tools. By using tools one can visualize as well as construct a complex geometry of the components. Hence, now a days, demands of this software is more. In the present work a 3-D image of quarter portion of flexible coupling is constructed in Pro-E Software, which is discussed below a) Construct a quarter circle of 100 mm radius. Then it is constructed for a hub length of 56 mm. b) Now, the quarter portion is ready. With reference to centre point quarter portion, draw quarter circle of size 30 mm. Using extrude command, this quarter circle is then extruded for a hub length. During the operation of extrusion a material is removed. c) The quarter portion is now ready for the generation of holes at respective points. There are two holes at the quarter portion. This is accomplished again by a extrude command of software. During the operation of extrusion a material is removed. A quarter portion of the flexible coupling is now ready for stress analysis.

3.2. *Import of a quarter portion of the flexible coupling to Ansys work-bench Sub-headings*

In this process, a 3D image of a quarter positions of flexible coupling is imported form Pro-E software to ANSYS software . This process is a prerequisite for a structure analysis of any machine component. The

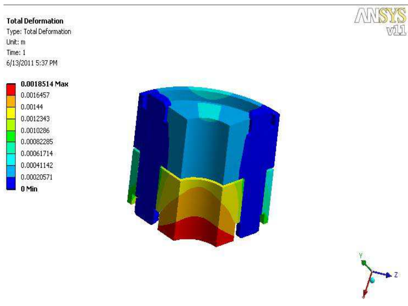


Fig.8

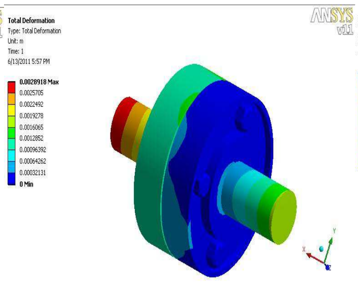


Fig.9

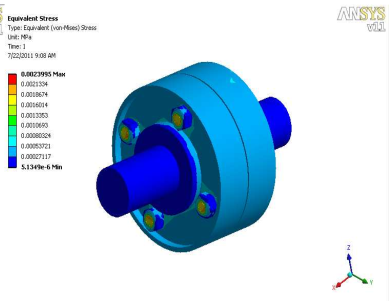


Fig.10

imported image of a quarter position of flexible coupling is considered for mesh generation. A tool of mesh generation helps for this process. This process required adequate time. After meshing, it is now required to apply boundary conditions and forces at respective positions of a quarter portion of the flexible coupling. Hence, for a quarter portions, fixed support is provided at respective points. Forces are also applied at respective position of bolts. Selection of material properties for a quarter portions for simulation process. In this section, desired values of material properties are choose. This material property is as Young's modules, Poissons ratio, density of material etc. The complete list of material properties for each material is given in material directory of analysis software. So, one can choose the required material properties directly from the list. Hence, for a quarter portions of a flexible coupling a material property for SAE 1030 material is chosen. Next to this, by using simulation process, the results of deflection conversely the corresponding a stress is estimated. The results of deflection of nodes and corresponding stresses are shown in Figure 8, Figure9, and Figure10.

4. DISCUSSION ABOUT RESULTS OBTAINED BY FEA APPROACH

The obtained displacement values, one can see the maximum displacement which is at node 1. This node 1 has a maximum displacement in X direction, whereas the minimum nodal displacement is obtained at node 10. The probable reason for maximum displacement at node 1 is discussed below.

(a) Node 1 is placed at the extreme corner of a 2D section of a quarter portion of the flexible coupling. (Kindly refer Figure 1)

(b) The Node 1 has one constraint at this point in Y-direction.

(c) When the Force F_1 is produced in a bolt, the Node 1 tends to deflect in X-direction. As refer to Figure 1 one can see that the position of node 1 is near to application of force F_1.

The probable reasons for minimum displacement are discussed with the following points.

(a) Node 10 is middle portion of a 2D section of a quarter portion of the flexible coupling. (Kindly refer Figure 1).

(b) When Force F_1 is produced in a bolt at node 2 at the same time, Force F_2 is produced due to reaction produced by a bolt on a flange body. Due to this action the deflection at node 10 is restricted.

The stresses induced in a flexible coupling by FEA approach is shown in Table 3. If one sees the maximum values of stress, then he may find the value which is at node 10. This value is of magnitude 1.1345N/mm^2. The probable reason to induce this stress at Node 10 is discussed below. When force F_1 acts in horizontal direction at the same time the reaction force F_2 acts vertically. Due to the action of these two forces, the node 10 bears more load. The next higher value of the stress is at node 9. This is because of the following reasons. The Node 9 is directly placed on the action of the forces F_1 & F_2.

5. CORROBORATION OF FEM RESULTS WITH ANSYS RESULTS

The estimated deflections by a FEA approach as well as by an ANSYS approach are compared in this article. The values of deflections at some nodes are less deviated but for some nodes, this deviation is considerable. The main intensive is that, how much percentage deviation in deflection values at all nodes. Hence, percentage deviation in deflection values at all nodes is estimated by using a formula, which is stated below. These values are shown in Table A1 (Appendix A).

Percentage deviation = (Max. deflection Value – Min. deflection value/Max. deflection value)

The average deviation is estimated as 18.71%. Hence, it is to be said that deflection at each node are well estimated through FEA Approach.

The most important and remarkable points of discussion is that, why there is a deviation in the values of deflections and stresses between these two approaches. This is discussed as below.

(1) When one selects, a FEA approach, then he has to consider two dimensional elements only. Because, it is quite difficult to solve a problem by considering 3D element.

(2) Hence, with a FEA approach, one can always obtained the results of any problem by considering 2D element.

(3) Whereas, if one put up an object for estimation of deflections and stresses in ANSYS Software then the software selects 3 dimensional elements which are more than a conventional FEA method.

6. CONCLUSION

After Through present investigation following conclusions are drawn, which are discussed below:

(1) When a high capacity motor induces a driving torque, it is then transferred to a cutter shaft through a coupling. The cutter shaft uses this torque for some useful work. But, at the same time, the process resistance offered by aluminium bloom provides a load torque on a cutter shaft. This load torque is then transferred through a coupling to the motor shaft. If load torque increases, conversely the motor produces additional driving torque to overcome this load torque. This load torque changes for every time-instance. During this action coupling becomes a node point, which sustains this transient dynamic action. Hence the coupling gets failed frequently.

(2) Using FEA approach, the following results are obtained the most sever point at which the load acted more is node 10. This node 10 bears high load as compared to other node. While node 9 is the next point, which bears the high load. Hence at node 10 and 9 the probability of crack formation is more.

(3) The deviation in ANSYS approach and the FEA approach are due to following reasons☹a)When one selects a FEA approach, then he has to consider two dimensional elements only. Because, it is quite difficult to solve a problem by considering 3D element. b)Hence, with a FEA approach, one can always obtained the results of any problem by considering 2D element). Whereas, if one put up an object for estimation of deflections and stresses in ANSYS Software, then it selects 3 dimensional elements, which are more than a conventional FEA method.

REFERENCES

[1] Hariharan V. And Srinivasan P.S.S. “Vibrational Analysis of Flexible Coupling by Considering Unbalance” World Applied Sciences Journal 8 (8): 1022-1031, 2010I SSN 1818-4952.

[2] Hariharan V. and Srinivasan PSS. “Vibration analysis of misaligned shaft –ball bearing system”. Vol.2 no. 9 (Sep 2009) ISSN: 0974-6846.

[3] Jackson, C., 1973, "Cold and Hot Alignment Techniques of Turbomachinery," Proc. 2nd Turbomachinery Symposium, Turbomachinery Laboratory, Texas A&M University, College Station, Texas, pp. 1-7.

[4] Sekhar, A.S., and Prabhu, B.S., “Effect of Coupling Misalignment on Vibrations of Rotating Machinery,” Journal of Sound and Vibration, Vol. 185(4), 1995, pp.655-671.

[5] Sinhaa.J.K. Leesb.A.W. , Friswellc.M.I. , “Estimating unbalance and misalignment of a flexible rotating machine from a single run-down”. Journal of Sound and Vibration 272 (2004) 967–989.

[6] Dewell dl and Mitchell ld (1984) “Detection of a misaligned disk coupling using spectrum analysis”. ASME Trans. J. Vibration, Acoustics, Stress & Reliability Design. 106, 9-16.

[7] Khot .S.M.,Pathan Shamim “Experimental investigation of effect of parallel misalignment in rotating machinery”. Proceedings of the NCNTE-2012, Third Biennial National Conference on Nascent Technologies Fr. C. Rodrigues Institute of Technology, Vashi, Navi Mumbai, Feb 24-25, 2012 Department of Mechanical Engineering.

[8] Kumar Sanjiv, Sehgal Rakesh, Kumar Rajiv, Bhandari Sanjeev, “ Vibrations Analysis of 4 Jaw Flexible Coupling Considering Unbalancing in Two Planes” International Journal of Science and Technology Volume 1 No. 11, November, 2012.

Appendix A

Table A1. Stresses of nine elements of 2d section of a quarter portion of A flexible coupling

Node Number	**Stresses (N/mm^2)**	**Node Number**	**Stresses (N/mm^2)**	**Node Number**	**Stresses (N/mm^2)**	**Node Number**	**Stresses (N/mm^2)**
Q1	-0.43905	Q4	0.303038	Q7	-0.1109	Q10	1.134546
Q2	-0.55163	Q5	-0.082351	Q8	-0.38713		
Q3	-0.237776	Q6	-0.061677	Q9	1.066516		

Table A2: Percentage deviation in deflection values at all nodes

Node. No.	**FEM Approach**	**ANSYS Approach**	**Deviation**
1X	0.0001775	0.0001423	19.83%
2X	0.0001734	0.0001446	16.60%
2Y	0.0000103	0.0000130	20.76%
3X	0.0000491	0.0000400	18.56%
3Y	0.0000097	0.0000070	17.83%
4X	0.0000592	0.0000501	15.55%
5X	0.0000523	0.0000675	22.86%
6X	0.0000645	0.0000432	23.02%
6Y	0.0000656	0.0000799	18.03%
7X	0.0000272	0.0000344	21.15%
7Y	0.0001530	0.0001130	16.14%
8Y	0.0001339	0.0001001	25.30%
9X	0.0000403	0.0000600	12.80%
9Y	0.0000098	0.0000070	15.57%
10X	0.0000286	0.0000400	16.50%
10Y	0.0000037	0.0000030	18.91%