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COMPARATIVE STUDY ON AUTOMATION OF GEAR MODELLING USING CATIA V5 (KBE) - VBA APPROACH

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ABSTRACT

The aim of this project is to work on automated modelling system for different types of gears with a new way of solving modelling design problems with in the CAD environment by using knowledge based engineering from the conventional design procedures to improve gear design phase in an industry. Computer aided design is an important industrial art extensively used in many applications including automotive, shipbuilding, aerospace industries and many more. Knowledge based engineering (KBE) has become a practical method of visualizing manufacturing cost and enabling product analysis by simulating product development activities in design for manufacturing support tools. Any of the mistakes in the design phase may result in to very high losses associated in the manufacturing investments. To reduce the development cost, it may be worthy to focus on the design phase of product development. By using CATIA V5 model of internal gear and external wobble gears have been created and then imported in ANSYS workbench for analysis. Static structural analysis has been conducted for both the gears by using ANSYS .The values of maximum stresses from structural analysis are compared with theoretical calculation. It is observed that 90% time will be reduced by using this automated gear designing and also chance of errors also reduces. A system has been developed to fill in the gap between design engineers and computational experts when analyzing product development process.

Keywords: catia, ansys, gears, knowledge based engineering, vba.

1. INTRODUCTION

The automotive industry is a major industrial and economical force worldwide. It makes around 60 million cars and trucks a year. In today's world, producing right product in right time and with high quality is a major criteria. From the very initial phase of a part till it ends, each and every stage is very important in an industry. Today's world is a customer driven world, where customer requirements are given an utmost importance to produce a product in the market. Satisfying customer requirements with good quality products is a major role. In today's market, it is very important to enhance the productivity. Now a days all most all the automotive companies use CAD/CAM/CAE as a tool to reduce the time span of design and manufacturing and completing these tasks. Computer-aided design is one of the many tools used by engineers and designers and is used in many ways depending on the profession of the user and the type of software in question. The CAD domain has always been an early adopter of software-engineering techniques used in knowledge-based systems, such as object-orientation and rules. Knowledge-based engineering integrates these technologies with CAD and other traditional engineering software tools. By using CATIA V5 we have created a model of internal gear and external wobble gears and then imported in ANSYS workbench for analysis. We have conducted static structural analysis of both the gears by using ANSYS .The values of maximum stresses from structural analysis are compared with theoretical calculation.

1.1 Literature survey

Akshaykumar V. Kadam [1] has done automatic assembly modeling using parametric modeling for different products. Reduction type gearbox manufactured by many companies across the world. Algorithm defined for the automatic assembly of the single stage helical reduction type gearbox. Algorithm is implemented with Pro/program in the generic assembly of gearbox. User interface is developed with excel VBA.

Gurjinder Pal Singh [2] automated the modelling system for different types of bearing and introduced a new way of solving the bearing design problems within CAD environment by using knowledge extracted from the conventional design procedures. This automated design tool helps in filling the gap between design engineers and computational experts when analyzing product development process.

Shubham A. Badka [3] has done analyses the bending stresses characteristics of an involute spur gear tooth under static loading conditions. The results obtained theoretically are in good agreement with those obtained from software. Also an attempt is made to introduce Stress and displacement characteristics of tooth under dynamic loading conditions.

Shubham A. Badkas and Nimish Ajmera [4] implemented parametric jewelry modeling in Visual Basic Applications (VBA) programming environment that runs simultaneously with AutoCAD and provides programmable control of AutoCAD through the ActiveX Automation interface. This paper presents a CAD paradigm that makes the technical contribution of developing a parametric jewelry modeler for designing traditional carved pendants.

Yashwant Mokhede *et al*, [5] automated modelling system for different types of bearing and new way of solving bearing design problems within CAD environment by using knowledge extracted from the conventional design procedures. This automated design tool helps in filling the gap between design engineers and

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computational experts when analyzing product development process. It can also serve as an aid in the manufacturing planning to evaluate manufacturability and life cycle simulation of product for cost estimation.

Yogesh H. Sawant and U. M. Nimbalkar [6] developed generation of 3D model of horizontal screw conveyor components in CATIA V5 software. The system automatically generates 3D model. The advantage of this developed system is that the end user did not need knowledge of complicated CAD software.

1.2 Significance of CAD automation

Before the start of CAD, the drawings are done in 2D design systems which are very difficult to understand. It is very hard to create, modify and identify interference & errors. Even for a small mistake the entire drawing should be recreated. 3D CAD software uses an intuitive and concrete way to present objects to users, which fundamentally avoids above mentioned problems.

In this project we discuss about how to integrateVBA macro with Catia which reduces the time and increases efficiency of a part. The logic for calculating independent variables are built into program and dependent variables will be calculated by relations with independent variables and design rules. Based on these dimensions and the relations between them, a 3D model of the gear will be generated by using visual basic macro.

The visual basic macro reduces the chance of errors/mismatch. Any changes in the design philosophy would entail modifications in the program and need to ensure proper changes in the individual designs would be eliminated. This would ensure better design control.

This automation of CAD will be very comprehensive and generates 3D CAD assembly and drawings for a multiple gears. This system helps the user without any knowledge in catia, by designing the part with menu driven prompts given by program instructor.

Automated modelling of spur gear is done by coding the formula in the macro. When the inputs are given in the graphical user interface the macro recognizes the inputs and the runs the code accordingly. These formulae are different for different gears. Macro reads the data given in the user interface and then updates the design. The following table 1 shows the calculation of spur gear using formula.

Table-1. Calculation of standard spur gears.

No.	Item	Symbol	Formula
1	Module	m	Set value
2	Reference Pressure Angle	a (Alpha)	
3	Number of Teeth	z	
4	Center Distance	a	(Z1+Z2)m/2
5	Reference Diameter	d	Zm
6	Base Diameter	db	d cos alpha
7	Addendum	ha	1.00m
8	Tooth Depth	h	2.25m
9	Tip Diameter	da	d+2m
10	Root Diameter	df	d-2.5m

All calculated values in table are based upon given module m and number of teeth z1 and z2. If instead module m, center distance and speed ratio i are given, then the number of teeth, z1 and z2, would be calculated with the formulas as shown in Table-2.

Table-2. Calculation of teeth number.

No	Item	Symbol	Formula
1	Module	M	
2	Center Distance	A	
3	Speed Ratio	I	-
4	Sum No. of Teeth	z1+z2	2a/m
5	Number of Teeth	z1, z2	i(z1+z2)+(i+1) / (z1+z2)/(i+1)

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2. DESIGN AND DEVELOPMENT

The design development is done on the basis of modelling and designing approach, where they are produced automatically with the help of macro. In modelling phase, the required inputs are given in the graphical user interface which gives the specific output automatically, after clicking the design button. Hence the modeling and designing are done simultaneously, which helps in reducing the time and chance of errors. The solution for design problem is generated on the basis of KB saved in the system. All the KB is saved in the form of VB code.

The design development starts type of gear which the user is going to design. After deciding which type of gear, then users have to proper inputs in the graphical user interface. The graphical user interface will be different for different types of gears since the inputs which are given are different. After giving the inputs the user has to run the macro and check whether the design is correct or not. If the design is not correct it then again goes to Inputs phase, there the user can change the inputs and can generate the design once again.

This is process flow of Automation of gear modelling using CATIA (KBE) - VBA approach. The Figure-1 represents flow chart for automated modelling gear design approach

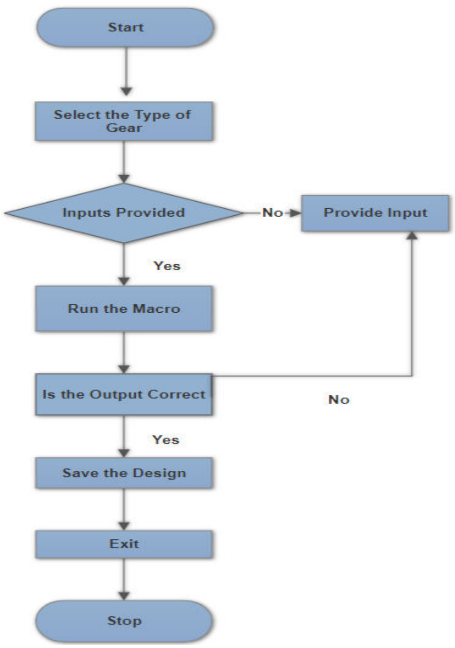

Figure-1. Flow chart for automated modeling gear design approach.

Step 1

User will have to run the project through VBA editor and run command button.

Step 2

User will have to give the user inputs to UI for required bearing design. After giving suitable inputs, user should click on the calculate command button to get the desired output.

Step 3

Calculation of gear will be done automatically on the basis of following formula.

Gear calculation for 20deg pressure angle:
No. Of teeth n=25
Module m=3.5
Pitch circle radius Rp=m*n/2
Clearance circle radius Rb=0.94*Rp
Addendum circle radius Ra=Rp+m
Dedendum circle radius Rd=rp-1.25*m
Fillet radius Rf=0.39*m
Angle = 90/N * 1deg

Step 4

All these calculated design parameters are then stored in to the database.

Step 5

All these calculated design parameters are then placed in to the HLCT (High Level CAD Templates) instead of respective variables and the program is recompiled to give output in CATIA as .CATProduct design of gear.

3. PARAMETERS DEFINITION

The complex geometry of the tooth recommends the generative shape design mode. In order to obtain a structured hierarchy, the tooth will be defined in a geometrical set - a definition set that relates intelligent geometric. The procedure will explained the following steps

Step 1

Before keying in the parameters, a wire frame has to be created with circle command in CATIA as a background. The 2D illustration of this design is shown in the Figure-3. Since the parameters, formulae and are all implemented using this background design. Figure-2: shows outer sketch of CATIA.

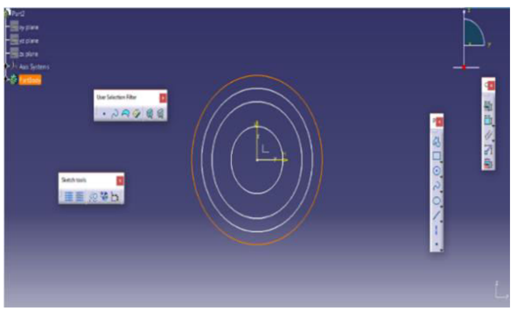

Figure-2. Outer sketch of CATIA.

Step 2

In the first step, will be defining the parameters such as modulus, no. of. teeth, outer pitch etc. the

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parameters are the connection between CATIA V5 and VBA macro. Figure-3 represents parameters definition.

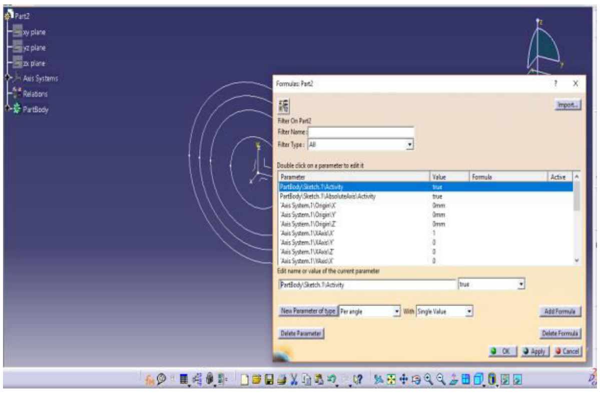

Figure-3. Parameters definition.

Step 3

The formulae in CATIA are coded with formulae button i.e., f(x).The involute spline curve will be the support of the tooth profile, design based on the defined parameters. The curve will be rotated against the ZX plane. The outer and root circle will be used for trimming, filleting, in order to finish the profile, which will be mirrored to complete a tooth. Figure-4 represents Construction of the tooth profile from parametric spline curve.

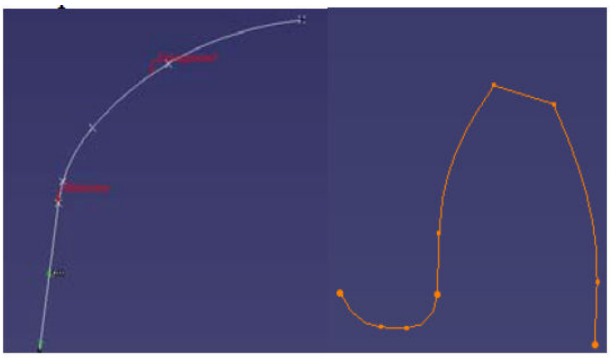

Figure-4. Spline curve of gear tooth.

The below Figure-5. Shows the example of 2D Illustration drawing of a spur gear for designing.

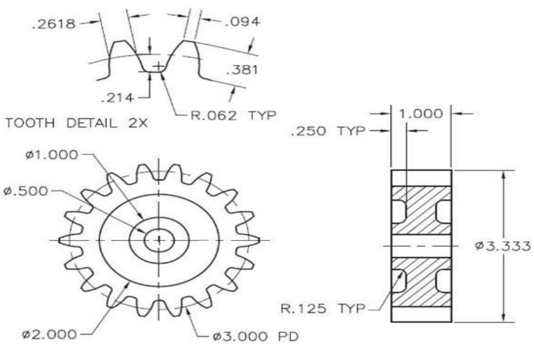

Figure-5. 2D Illustration drawing.

Step 4

The tooth profile will be arrayed, joined for a sketch and extruded in part design mode as a basic Pad feature will be obtained. Figure-6 shows the CAD for spur Gear.

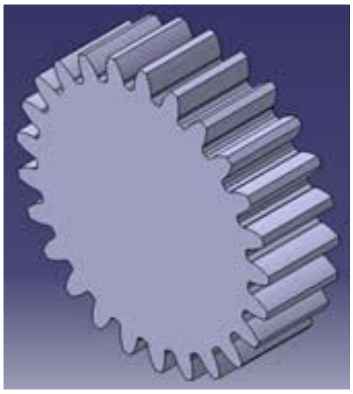

Figure-6. CAD Spur gear.

Step 6

After that we do a manual process where we constraint and simultaneously add angle, radius etc. Then the parameters appears with part design. Figure-7 represents manually inserting parameters.

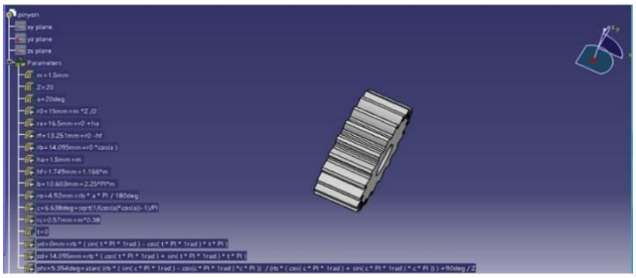

Figure-7. Manually inserting parameters.

Step 7

Once the formulae is added and set up is done, from next time onwards we can change any value in the parameters, update it, then we can see the modified design. Figure-8 new 3D CAD of spur gear.

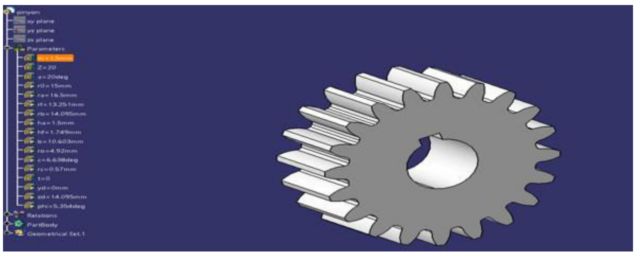

Figure-8. New 3D CAD.

4. RESULTS AND DISCUSSIONS

System gives the solution for industrial real time problems by using knowledge based engineering (KBE) approach. In this approach loading conditions and material property (strength) is taken as input. By using KB saved in

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the system and input parameters, optimum design solution is calculated as per the formulae provided in to the system and 3D CAD assembly solution is directly given in to the CATIA V5- R19.

Spur gear, helical gear and worm and worm wheel gear these are the three different types of gears which can be created by using the developed system. As discussed, the inputs should be given to the UI which then develops the 3D Cad geometry automatically within few seconds in CATIA V5.This automation is helpful for the user who is illiterate on CATIA as well as in VB.net. User hasto make sure CATIA application is opened before running the UI. The output and user interface for different gears are shown in as below.

4.1.1 Spur gear

For spur gear, we provide no. of. teeth, modulus (m) and circular thickness (T) these are the inputs taken from the user through UI as shown in Figure-9 shows user interface of spur gear and Figure-10 shows the 3D CAD output.

Figure-9. Graphical user interface of spur gear.

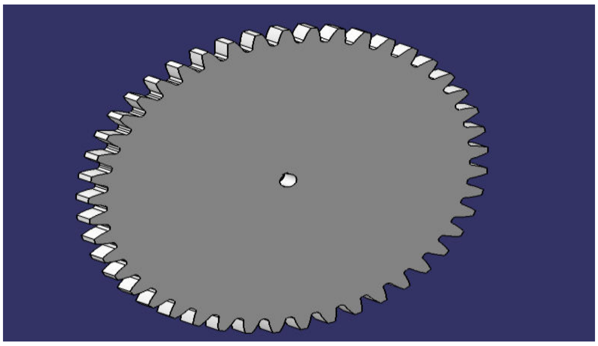

Figure-10. Output of 3D CAD.

Similarly the modelling is developed for bevel gear and worm gear.

4.2 ANALYSIS OF GEARS

For finite analysis of gear only teeth section is considered. The total analysis is done in ANSYS workbench.

The final mesh is generated. The mesh is then exported as solver deck for Ansys workbench.

- element size : 0.8mm
- mesh is done using hexa elements
- total number of elements in model: 88600
- total number of nodes in the model : 101062

Properties of material used are shown in Figures11 and 12.

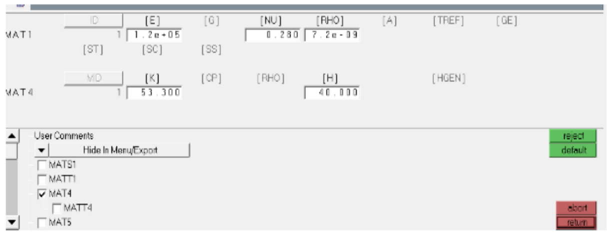

Figure-11. Properties of steel.

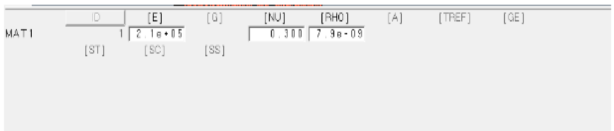

Figure-12. Properties of grey cast iron.

4.2.1 Structural analysis (boundary condition)

The center nodes are constrained with zero degrees of freedom. Figure-13 represents the center nodes

A force 1000N is applied normal to the gear teeth. Figure-14 shows force acting on the gear.

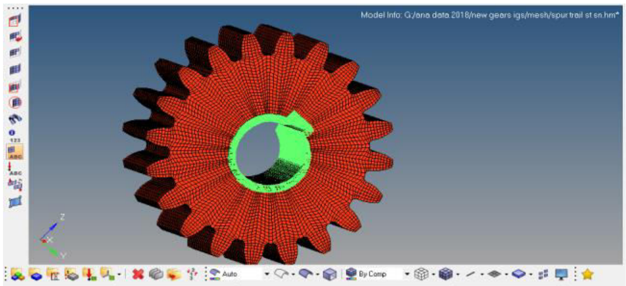

Figure-13. Center nodes for structural analysis.

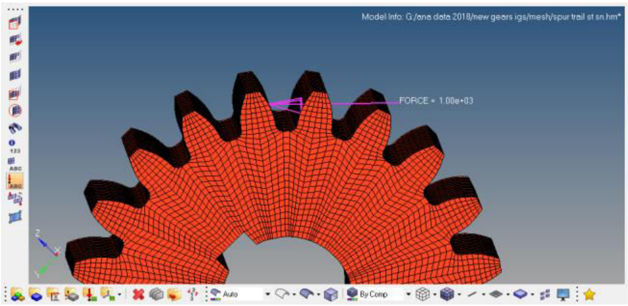

Figure-14. Force acting on the gear.

Final view after applying boundary condition. Figure-15 shows boundary condition.

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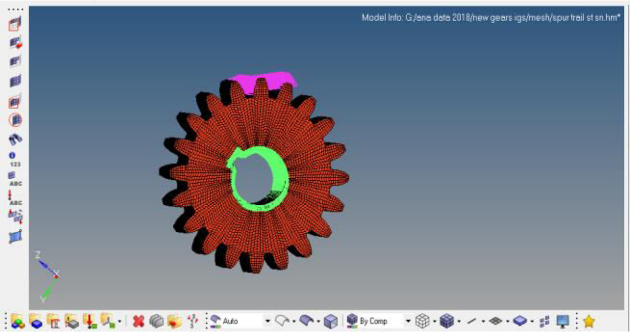

Figure-15. Boundary condition.

4.2.2 Thermal analysis

Nodes are constrained at the centre. Figure16 represents the center nodes

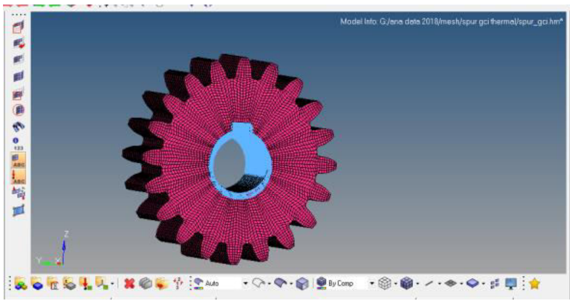

Figure-16. Center nodes for thermal analysis.

A flux of 2.5W/m^2 is made to act on the gear teeth. The flux is created due to friction between mating gears generation heat which is to be analyzed. Figure-17 shows thermal heat flux acting on a gear

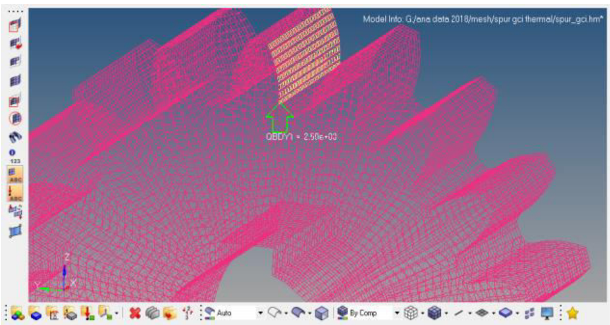

Figure-17. Thermal heat flux acting on a Gear.

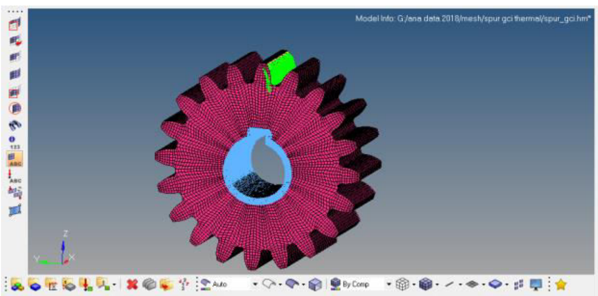

Figure-18. Final mesh model.

4.3 FINAL RESULTS

4.3.1 Spur gear

4.3.1.1 Stress analysis for grey cast iron

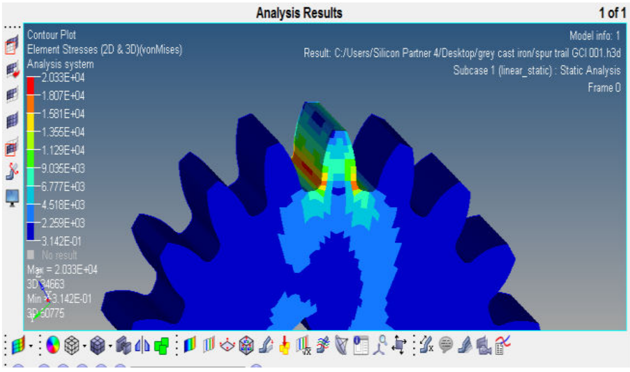

Figure-19. Stress - static analysis for grey cast iron.

4.3.1.2 Displacement analysis for grey cast iron

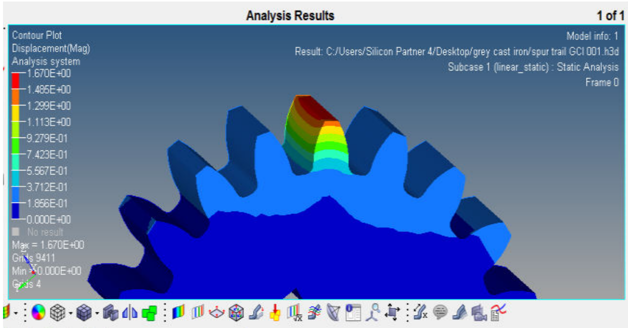

Figure-20. Displacement- static analysis for grey cast iron.

4.3.1.3 Stress analysis for steel

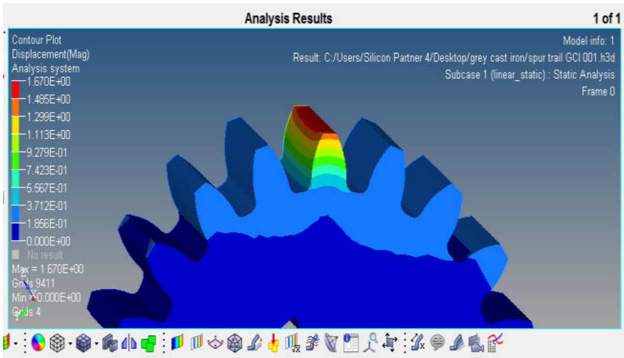

Figure-21. Stress-static analysis for steel.

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4.3.1.4 Displacement analysis for steel

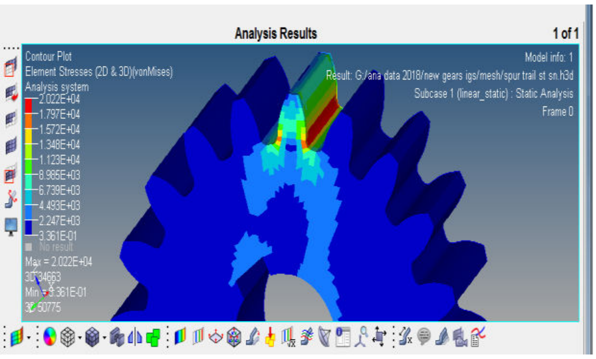

Figure-22. Displacement - static analysis for steel.

4.3.1.5 Thermal analysis flux

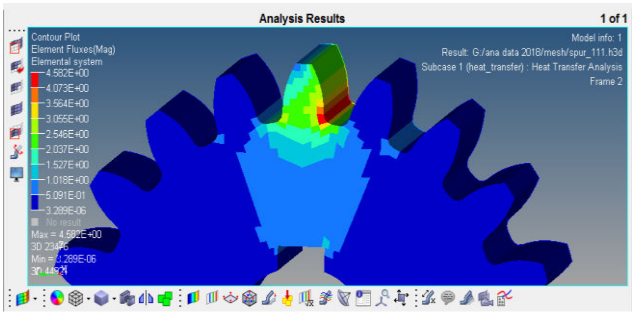

Figure-23. Flux.

Temperature distribution gradient:

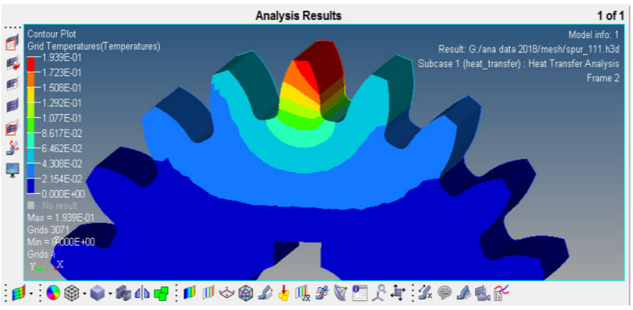

Figure-24. Temperature distribution gradient.

The analysis shows that force acting will be the same and if there any difference it is very too easy regenerate the model and do the analysis

4.4 Advantages of Macro

With the help of this macro user without catia knowledge can easily design the gear. This macro has inbuilt with some features in such a way that it won't allow any errors to get happened during the designing.

The advantages of this macro are.

- A gear should have maximum of 200 teeth. This automation stops the design if user enters no. of. teeth more than 200. It helps the user to reduce chance of making an error while designing a gear. Figure-1 shows the constraints on no. of teeth in graphical user interface.

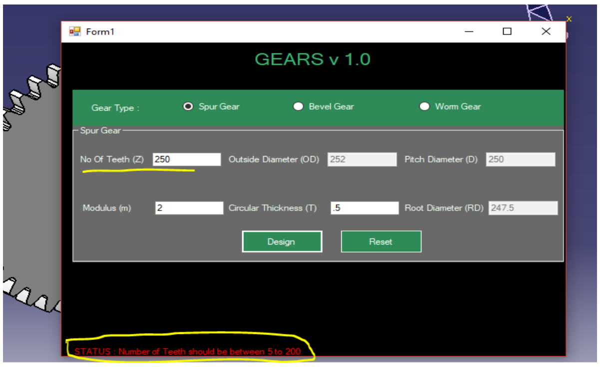

Figure-25. The constraint of no. of teeth in graphical user interface.

- The circular thickness of a gear should be 0 - 1. If User enter the thickness value more than 1 it shows an error saying that "circular thickness should be 0 to 1".This notifies the user to change the value and then generate the design. Figure-26 shows the constraint of circular thickness in graphical user interface.

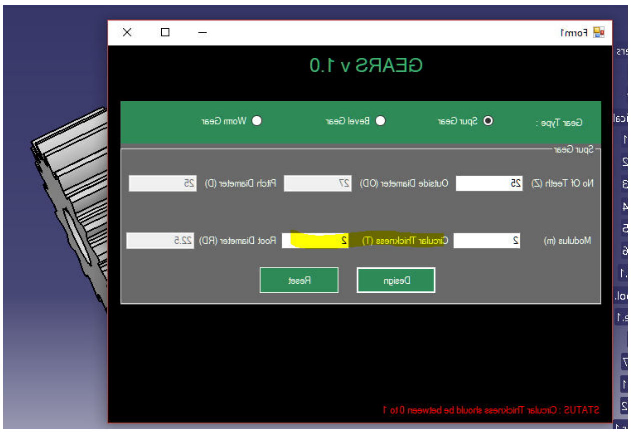

Figure-26. The constraint of circular thickness in graphical user interface.

- The module of a gear should be greater than 1. If user enters the value less than 1 the macro will not generate the model. It determines that user has entered the wrong value.
- All the text boxes shown in graphical user interface should be numeric.

5. OBSERVATION

Following Table-1 gives the average time required for manual modeling done by any proficient expert in CATIA and automated modeling done by using system developed in this work. It indicates that the required time for automated modeling is very less than the manual modeling with the similar accuracy in both approaches also the design calculation and selection of appropriate standard size for coupling is automatically done within few seconds in automated design approach.

Table-1. Average time for manual modelling and automatic modelling.

Types of gears	Manual modelling	Automated modeling
Spur Gear	2-3 hrs.	30 sec-1min
Bevel Gear	2-3 hrs.	30 sec-1min
Worm Gear	3-4 hrs.	30 sec-1min

Following Table-2 gives the average time required for manual modelling done by changing one of the parameters and automated modelling using macro. The below table shows the time taken for each gear for manual modelling and automated modelling when there is an change in parameter like no.of. teeth, modulus. From this table it can be observed that 90% time will be reduced by using this automated gear designing and also chance of errors also reduces.

Table-2. Average time taken for change in parameter manual modelling and automatic modelling.

Types of gears	Change in parameter	Manual modelling	Automated modelling
Spur Gear	No.of teeth = 55	2 hrs.	30 sec-1min
Bevel Gear	Modulus = 4	2 hrs.	30 sec-1min
Worm Gear	Modulus = 4	3.5hrs.	30 sec-1min

6. CONCLUSIONS

A system has been developed to fill in the gap between design engineers and computational experts when analyzing product development process. This system is developed in two approaches one is automatic modeling and to give design solution for gear design. This study can be extended up to building in geometric data and mechanical elements libraries, using generative structural analysis module or for editing the graphic interface and new functions definition, using microsoft visual basic, for easy geometry insertion in the model space.

The following conclusions can be drawn from this study

a) Spur gear, bevel gear and worm gear, are the three different types of gears used for modelling.
b) This design development gives automated design solution for required customer specifications i.e., no. of teeth, modulus, pitch etc.,
c) Automation of 3D CAD using VB.net helps in reducing manual errors during modelling as user has to give inputs in a single graphical user interface without using any commands in CATIA

This newly defined function can integrate automatic calculus of the assembly features, maximum torque etc. This study can be extended to bevel gears, and more detailed geometry (structural holes or gear base).

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