Parametric Skeleton Based Top-down Assembly Design Implementation with a Simple Illustration

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Abstract Based on object oriented program design and waterfall application design methodology in software field, mechanical designers extensively using CAD software started implementing similar data and information organization in mechanical design. After testing different ways, the mechanical CAD design fraternity adopted 'top-down design' term for method which can be implemented for robust CAD designs which are easier to maintain and alter. CAD softwares provided some tools within the CAD environment to implement these principles at the discretion of the designer. Top-down design methodology can be implemented using parameters or geometry sharing. In this work, the first method is discussed with a simple example of a protected type flanged coupling. The design intent including the basic equations can be embedded into the CAD design with careful use of parameters and top-down tool. The use of relations allows building dimensions common to two or more parts and allows selection and assembly of proper bought-out parts like keys, bolts etc. Modifying a key parameter percolates the changes into all the components and regenerates the design while maintaining the design

Keywords top-down design, parameter, relations, protected type flanged coupling

1. INTRODUCTION

A top-down design consumes more time up-front at concept stage for designing efficient data sharing skeletons. The detailing of concepts in terms of parts, sub-assemblies & assemblies can begin only after skeletons at all levels have been designed. Hence, lots of designers assume that top-down design is too cumbersome and time-consuming to use and unnecessarily delays the delivery of production drawings. But considering the time spent in modifying the models to meet the design changes, design models based on top-down design are much more robust & fast to regenerate as compared to models built by any other method. At the end of a design cycle, therefore, systems built with top-down design will have required less time and end up being more robust. CREO is a parametric modeling software which provides some top-down design tools which mainly include use of skeletons &/or layouts & reference control. These tools can be effectively used with designer's discipline for top-down design in two ways a) Parameters governing the geometry are determined in the skeleton & shared to respective parts from the skeleton to govern the design. b) Geometric entities like curves, surfaces etc. and datum entities like point, axes, planes etc. are created in skeleton which are representative of form & placement of the parts & these geometric entities are shared to respective parts from skeleton to govern the design. The skeleton entities are fully parameterized.

2. PROTECTED TYPE FLANGED COUPLING

A typical assembly of protected type flange coupling mainly consists of the driving & driven shafts, two flanges, two keys, certain number of bolt-nut pairs depending on the design. A half cross sectional of the assembly identifying the important dimensions is provided in Fig.1.

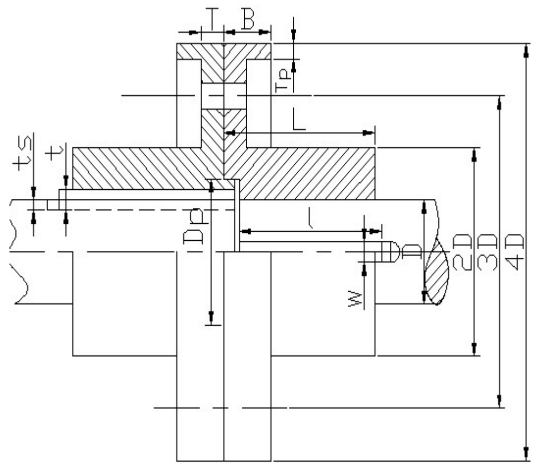


Fig.1 Half Sectional View of a Typical Protected Type Flanged Coupling with Important Dimensions

The basic design problem data considered for parametric design are listed hereunder.

Power : 40 KW,
RPM: 450,
Design Torque = 1.25 * Rated Torque,
Safe Shear Stress for shaft, key & bolts : 40 N/mm^2,
Safe Direct Stress for Flanges : 80 N/mm^2,
Safe Shear Stress for Flanges : 15 N/mm^2

3. PARAMETRIC CAD DESIGN

The skeleton model in assembly mode is first created with parameters and relations. These are then shared to the parts in the assembly mode. The parameter definitions in the CREO model for important dimensions of the flanged coupling are listed in Table-1. The complete design is carried out in assembly mode.

3.1 Design of Skeleton Model

The main assembly is created & the default skeleton in the CREO assembly is inserted. Various parameters required for governing the different dimensions of various parts are created in the skeleton using the parameters interface. The design relations are built as shown in the screen capture of the relations tab of the skeleton in Fig.2. The empirical relations design are adopted rather than cluttering the relations with design checks & confusing with the design equations which will lead to other direction than focusing on the concept of top-down design. The empirical design relations as encoded in Fig. 2 are obtained from widely respected and celebrated domain references like [4].

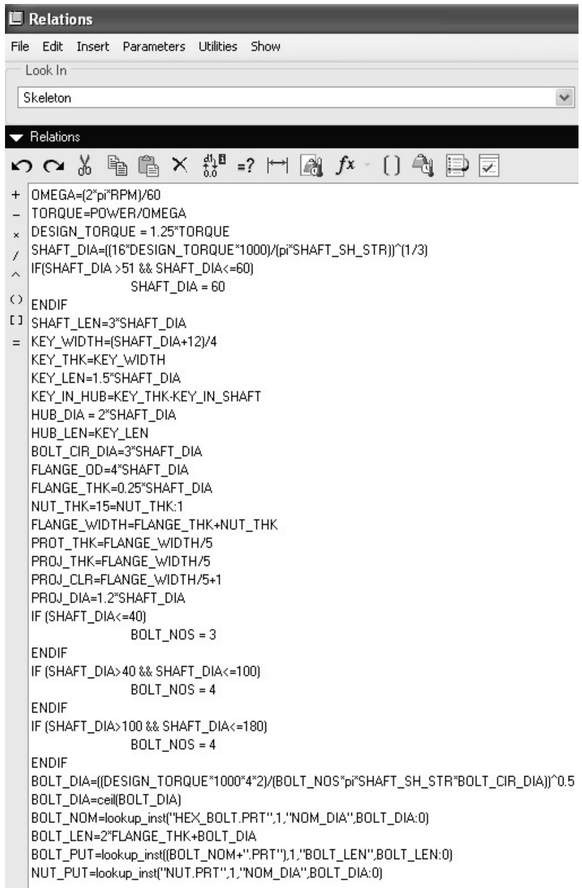


Fig.2 Screen Shot of the Relations Tab of the Skeleton Model with Design Relations

Table 1 Dimensions & Corresponding CREO Parameters

Dime-nsion	CREO Parameter	Description
	POWER	Power Transmitted
	RPM	Revolutions Per Minute
	OMEGA	Angular Rotational Speed
	TORQUE	Torque
	SHAFT_SH_STR	Shaft Shear Stress
D	SHAFT_DIA	Shaft Diameter
	DESIGN_TORQUE	Torque for Design
	SHAFT_LEN	Shaft Length
W	KEY_WIDTH	Key Width
T	KEY_THK	Key Thickness
L	KEY_LEN	Key Length
2D	HUB_DIA	Flange Hub Dia
L	HUB_LEN	Flange Hub Length
3D	BOLT_CIR_DIA	Flange PCD for Bolt Holes
4D	FLANGE_OD	Flange Outside Dia
T	FLANGE_THK	Flange Thickness
	NUT_THK	Nut Thickness

B	FLANGE_WIDTH	Flange Width
Tp	PROT_THK	Flange Protection Thickness
	PROJ_THK	Projection Thickness
	PROJ_CLR	Projection Clearance
	BOLT_NOS	Bolt Numbers
Dp	PROJ_DIA	Projection Diameter
	BOLT_DIA	Bolt Diameter by Design
Ts	KEY_IN_SHAFT	Depth of Keyway in shaft
	KEY_IN_HUB	Depth of Keyway in Hub
	BOLT_NOM	Nominal Bolt Size
	BOLT_LEN	Length of Bolt
	BOLT_PUT	Actual Bolt Selected from Family Table
	NUT_PUT	Actual Nut from Family Table

Once the parameters & relations are properly created the parameter report is exported to a ".csv" file typically appearing as indicated in Table-2.

Noting the fact that a coupling is axis-symmetric, a datum axis is created in the assembly for assembly of components as indicated in Fig. 3.

Table 2 Skeleton Parameters and their Details

Name	**Value**	**Access**	**Source**
POWER	40000	Full	User-Defined
RPM	450	Full	User-Defined
OMEGA	47	Locked	Relation
TORQUE	848	Locked	Relation
SHAFT_SH_STR	40	Full	User-Defined
SHAFT_DIA	60	Locked	Relation
DESIGN_TORQUE	1061	Locked	Relation
SHAFT_LEN	180	Locked	Relation
KEY_WIDTH	18	Locked	Relation
KEY_THK	18	Locked	Relation
KEY_LEN	90	Locked	Relation
HUB_DIA	120	Locked	Relation
HUB_LEN	90	Locked	Relation
BOLT_CIR_DIA	180	Locked	Relation
FLANGE_OD	240	Locked	Relation
FLANGE_THK	15	Locked	Relation
NUT_THK	15	Locked	Relation
FLANGE_WIDTH	30	Locked	Relation
PROT_THK	6	Locked	Relation

PROJ_THK	6	Locked	Relation
PROJ_CLR	7	Locked	Relation
BOLT_NOS	4	Locked	Relation
PROJ_DIA	72	Locked	Relation
BOLT_DIA	10	Locked	Relation
KEY_IN_SHAFT	7	Locked	Relation
KEY_IN_HUB	11	Locked	Relation
BOLT_NOM	M10	Locked	Relation
BOLT_LEN	40	Locked	Relation
BOLT_PUT	M10_40	Locked	Relation
NUT_PUT	HEX_NUT_M10	Locked	Relation

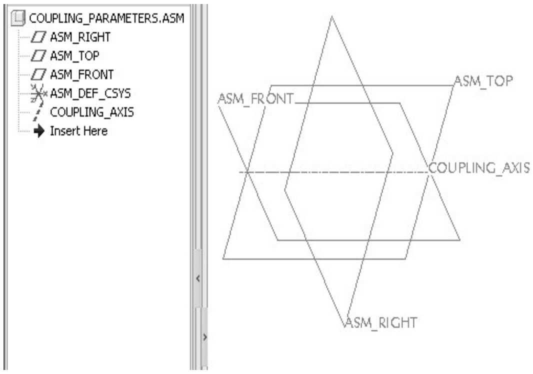


Fig. 3 Screen Shot of the Assembly Model Window with Coupling Axis Identified

3.2 Design of Part Models and their Placement

Thereafter the corresponding parameters from the skeleton are to be transferred to the parts where these parameters need to be used by means of cross part relations within the subassembly. For this purpose the reference control may be set to sub-assembly mode as shown in Fig. 4.

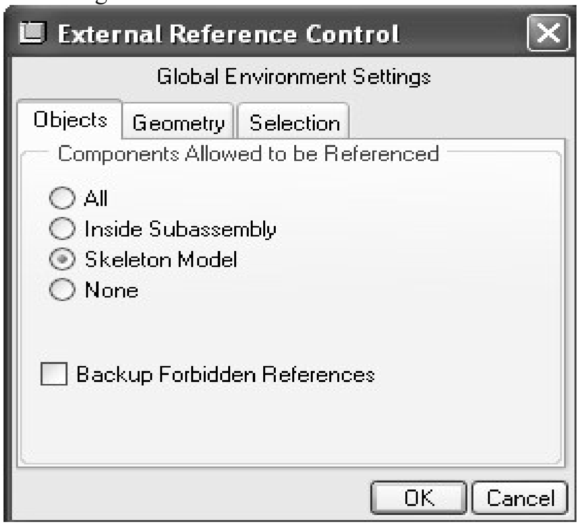


Fig. 4 Setting Reference Control for Information Sharing

Required parameters are created locally in the part models and these local part parameters are related to the skeleton parameters in the part relations tab using the session id of the skeleton (which in this case is 0) as demonstrated for the shaft part in Fig. 5.

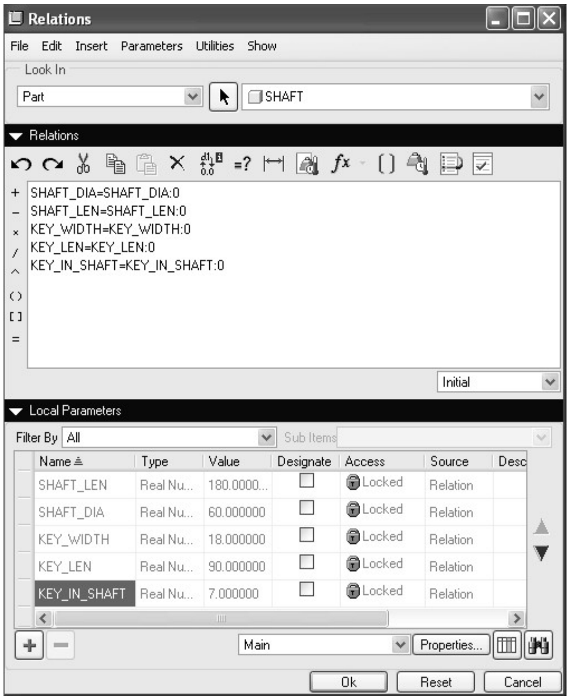


Fig. 5 Relating Local Parameters with Skeleton Parameters for Shaft Part

The part sections are controlled using the local parameters. The section dimensions are related to the local parameters either through sketches, features or datum entities. The dimensions of the keyway in the shaft are being governed by local parameters through relations provided in the sketcher itself as shown in Fig. 6.

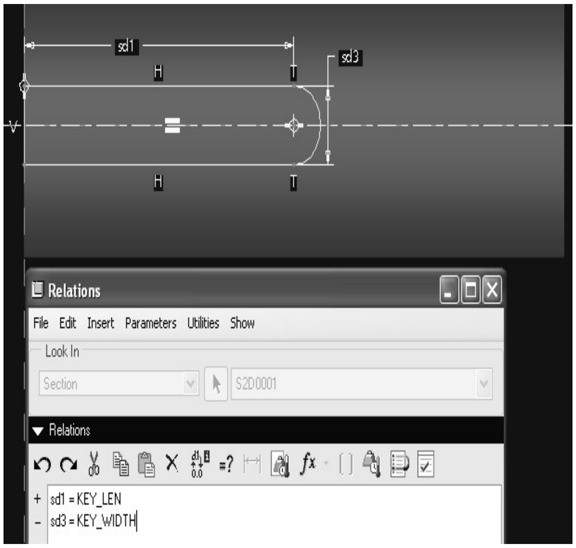


Fig. 6 Governing Part Sections, Features and Entities by Local Parameters

3.3 Designed Parts with Family Tables

Similarly the part relations are setup for the flange part as well. Flange part is converted into a generic part for a family table to address the location projection & recess. The generic must have both features & corresponding features are turned on or off in the family table members as indicated in Fig. 7.

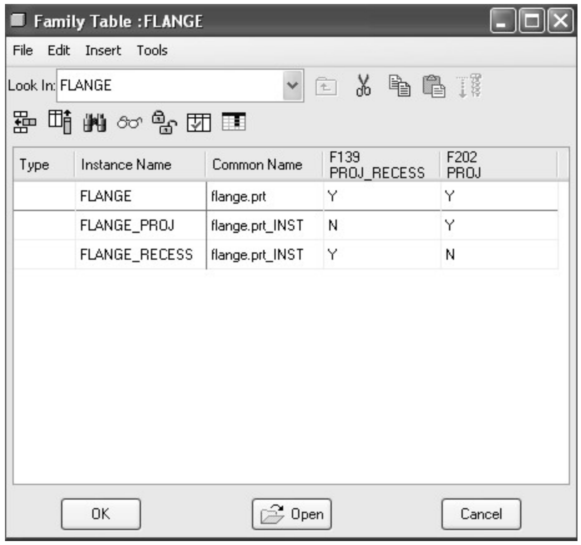


Fig. 7 Appropriate Features Turned on or off in Family Table Members for the Flange Part

The shaft part & flanges are assembled in position in the assembly using the axis created earlier. The keyways for the shafts are offset by a right angle to avoid gross weakening of the assembly as indicated in Fig. 8.

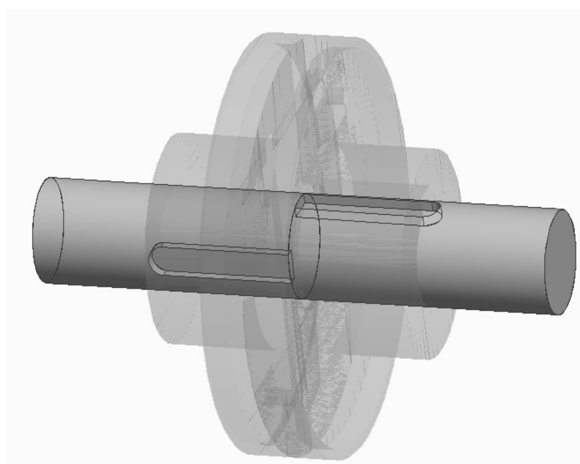
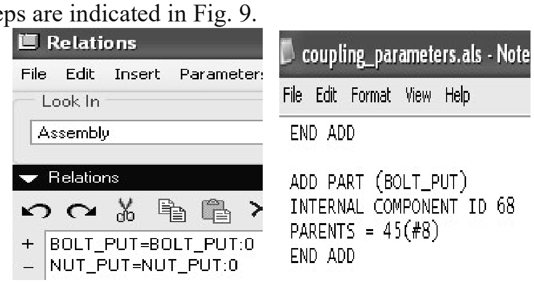


Fig. 8 Assembly of Shafts and Flanges with Proper Position of Keyways & Bolt Holes

3.4 Family Table based Standard/Library Parts

Bolts & nuts are standard, catalogue or library parts which are not designed to the assembly. These need to be selected from a standard repository or library of components based on the nominal size. The lookup_inst() function is used to retrieve the correct name of the bolt & nut based on the design evaluation of the bolt size. These names are assigned to the parameters created in assembly environment. The assembly pro-program has to be edited to insert proper bolt & nut parts. These steps are indicated in Fig. 9.



(a) (b)

Fig. 9 (a) Inserting Assembly Relations (b) Assembly Program for Proper Bolt Instance

Same procedure is adopted for the nut. The key may be treated as a standard component or designed to meet the assembly design. This is the manufacturer's decision. Here, it is treated as a designed component.

On completion of the design with these steps and successfully regenerating the assembly, the assembly appears as indicated in Fig. 10

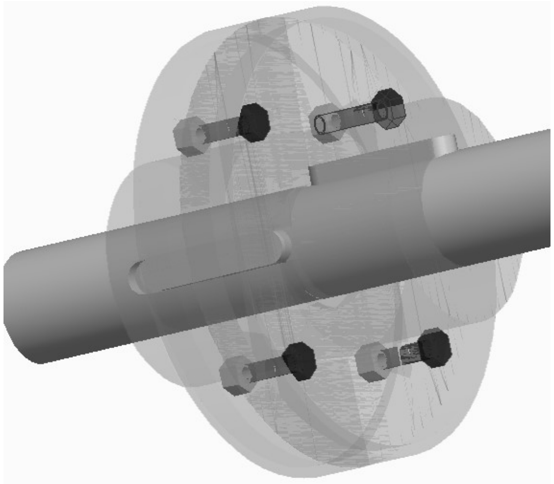


Fig. 10 Completed Assembly Model

4. CONCLUSIONS

Now to understand the benefits of this procedure, let us change the design data. The power is changed to 20KW and RPM to 600. Simple regeneration of the assembly provides the new design as indicated in Fig. 11. This can be created as a family table member or as a copy of the original design to preserve the drafts of the original drawing & have the same generated for the new one.

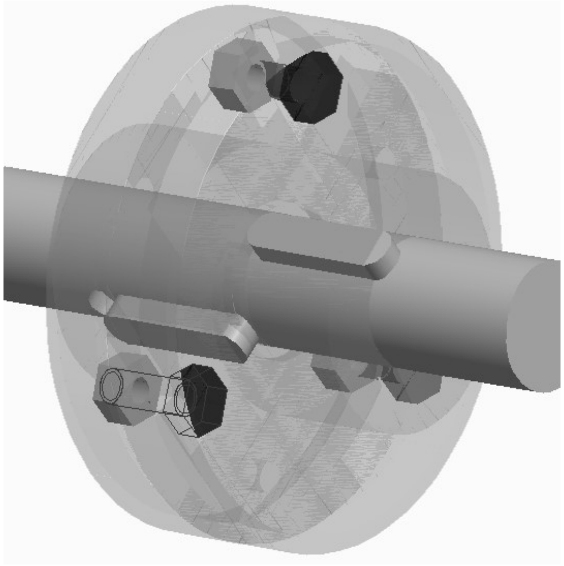


Fig. 11 Regenerated Assembly for Modified Design Data

An integrated approach for product design can also help in automated generation of bills-of-materials. Parametric method is a method to gain parametric relations across far-reaching models from a single interface while at the same time minimizing parent/child relationships. Although it requires a little upfront planning, and sometimes duplicate work of creating geometry and relations, parametric top-down approach offers some important advantages:

- huge reductions in time spent tweaking data in several models
- rapid alteration of models and design intent from a minimum of input points
- documentation of design intent, and more efficient use of computer hardware.

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