F2014-LWS-024

EARLY STAGE OPTIMIZATION OF CRANKSHAFT MASS USING DESIGN ASSISTANCE SYSTEM (DASY)

Bogomolov Sergii*; Macek Jan; Mikulec Antonin; Novotny Tomas; Kazda Josef;
Czech Technical University in Prague, Czech Republic

KEYWORDS – crankshaft, optimization, FEM, DASY, genetic algorithm

ABSTRACT –
This paper presents an optimization of crankshaft mass using Design Assistance System (DASY) software. The target of this optimization was to modify crankshaft design in order to obtain minimal possible mass while keeping stresses at acceptable level. Main crankshaft dimensions were not changed to allow usage of optimized crankshaft in the engine where original crankshaft was used.
In order to perform such optimization, a real crankshaft was measured and its full detailed model was created. Then from this full model a simplified model was created to reduce computational time and make optimization possible. Optimization was performed on the simplified model and results were checked on the full model to confirm their validity. Finite element models were created and solved in ABAQUS software. Parametric models were changed using script written in Python language. Optimization process was done and supervised by DASY. DASY was used to define optimization parameters and targets, to automate model modification and solution, and to run the optimization process. A Pareto front was obtained as a results of this optimization. This front represents the best compromises between crankshaft mass and maximal stresses. This means that each solution represents a design with the smallest possible mass for some level of stresses. For different level of stresses there will be different minimal possible masses.

TECHNICAL PAPER –

MODEL IN ABAQUS

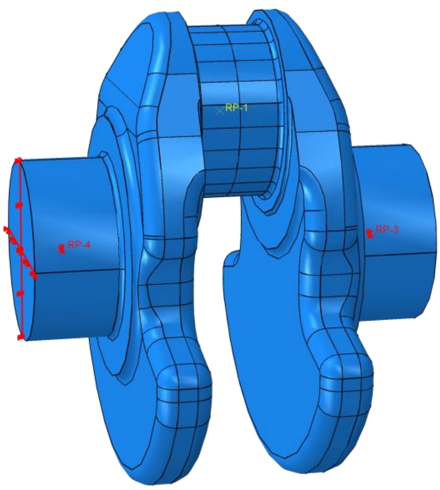

Fig. 1. Parametric model of crankshaft in ABAQUS.

Crankshaft is one of the most critical and important parts of the internal combustion engine (ICE) and its proper design is an important task. A real crankshaft was measured and modelled in ABAQUS software (Fig. 1). [4] This model is quite detailed, has many elements (Fig. 2 and Fig. 3) and is good for one-time computations. But for optimization purposes a simplified model which can be computed reasonably fast is needed.

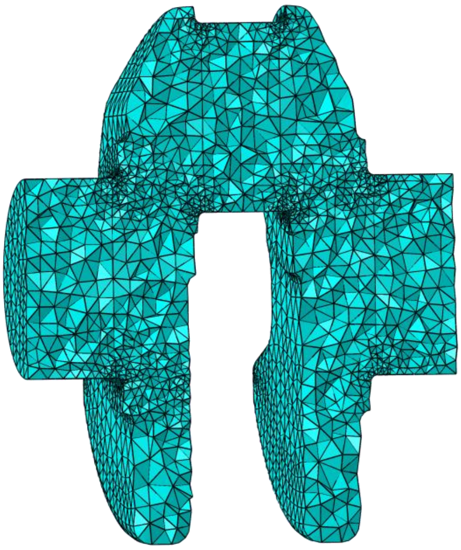

Fig. 2. Meshed model of crankshaft in ABAQUS.

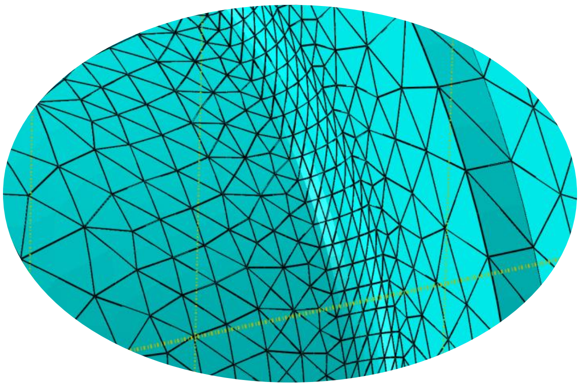

Fig. 3. Mesh in stress concentration region.

In order to reduce computational time the number of elements must be reduced. One approach is to increase the size of element. This will decrease accuracy of the results, but we can always verify final design using more accurate model. Second approach is to use only part of the model with corresponding boundary conditions to define planes of symmetry. In this study combination of both approaches was used.

Model of crank has two planes of symmetry as shown in Fig. 4 and Fig. 5. Boundary condition to represent the plane of symmetry restricts displacement in the direction of normal to the plane of symmetry and all rotations except the rotation around the normal to the plane of symmetry.

Another simplification that can be done is to use linear elements instead of quadratic ones. All listed simplifications have led to a model that can be computed in 21 second compared to 12 minutes for the original model on the same machine.

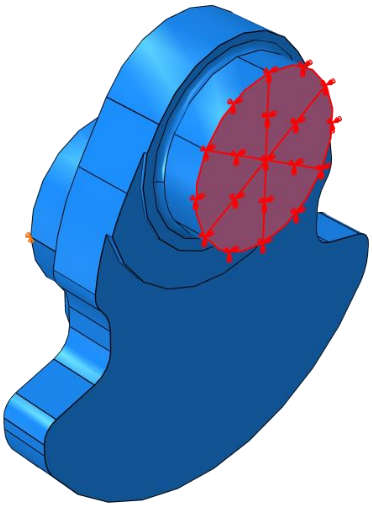

Fig. 4. First plane of symmetry for crank.

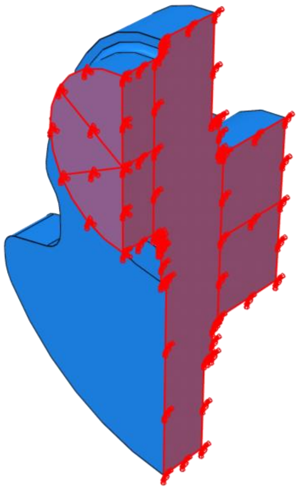

Fig. 5. Second plane of symmetry for crank.

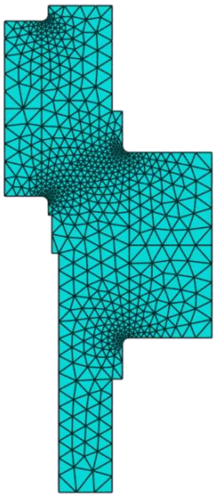

Fig. 6. Simplified finite-element model.

To automate the process of parameters update and model computation, a script in Python programming language was written. Python is natively supported by ABAQUS, which allows direct interaction with the model.

DESIGN ASSISTANCE SYSTEM

Design Assistance System (DASY) was used for crankshaft mass optimization (Fig. 7 and Fig. 8). The tool at the current state of development is a computer program with which one is able to create and maintain a knowledge database [1]. It is being developed using C# programming language [2] and Microsoft .NET Framework [5]. The main goal is the high level of flexibility along with a simple user interface. In this paper only automotive application is considered, but the software itself is not limited to this area. The key features of this tool are:

- creating components, assemblies and structural requirements,
- defining parameter dependencies via various methods,
- storing, grouping and reusing created components,
- storing, grouping and reusing created models,
- exporting geometrical data to CAD systems,

- multi-objective optimization.

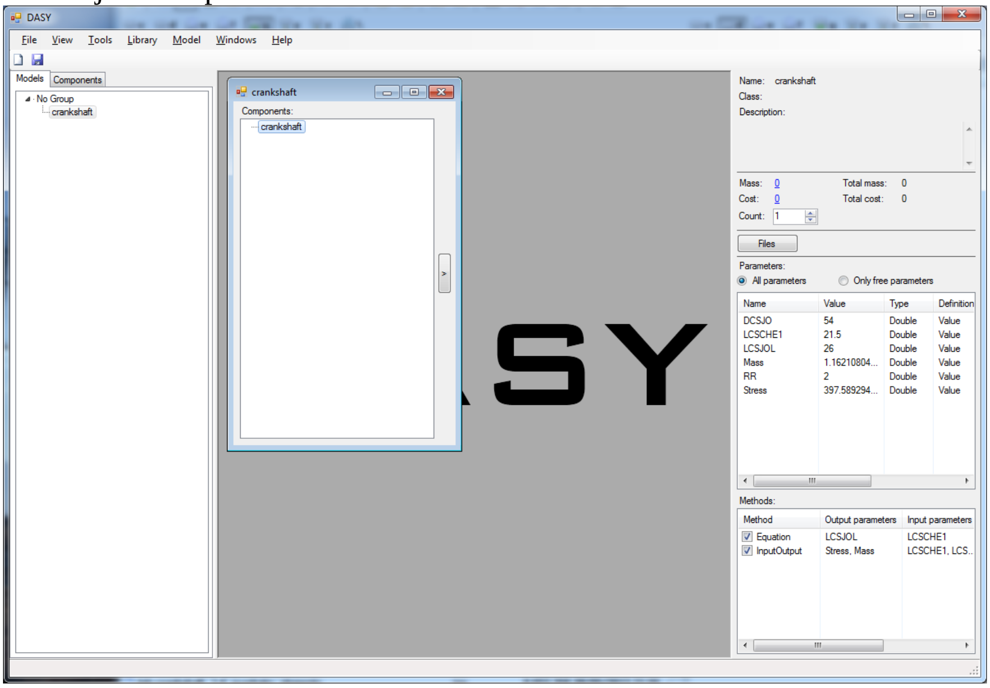

Fig. 7. DASY main window with loaded crankshaft model.

The tool is responsible for two major tasks. The first one is to provide a flexible knowledge database management tool. The second one is to provide an engineer with a framework to interact with a number of external software along with implemented algorithms and automate these interactions to provide a possibility of design optimization.

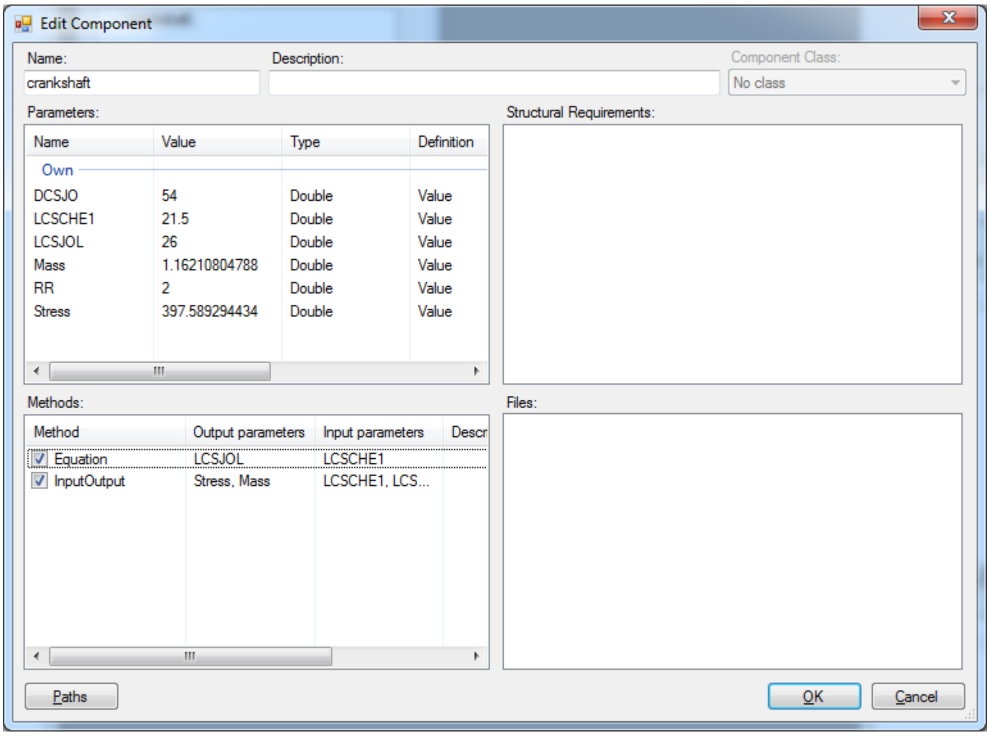

Fig. 8. Crankshaft component with parameters.

The first task requires storage and management of design data. Current task is complicated by required high level of flexibility. This means that fixed database structures cannot be used. It was decided to store data in Extensible Markup Language (XML) format. XML is a set of rules for encoding documents in machine-readable form. It is defined in the XML 1.0 Specification [6] produced by the World Wide Web Consortium (W3C), and several other related specifications. The design goals of XML emphasize simplicity, generality, and usability. It is a textual data format which allows editing by hand. Although the design of XML focuses on documents, it is widely used for the representation of arbitrary data structures.

It is also important to save the path to the final design, so it will be possible for successors to understand what decisions were made and why. This can be done by saving intermediate versions of designs and using commentaries. In the tool, any parameter, component, method or model can have a description that may explain why some approach is used or why some modification was made.

Second task requires a general transparent interface for end user to use external software from DASY framework. This is done through methods approach. Method is a transfer function from input to output parameters. At the current stage of development, following methods are implemented in the tool:

- Equation method defines a relation between parameters using simple algebraic equations.
- Input & Output Files method interacts with external software using input and output files approach.
- Description method allows entering some text description, for example name of the expert who can be asked about parameter values, or literature sources.
- CAD Model is also a method that allows updating of CAD model. This method utilizes an implemented plugins system, which means that there is no difference what CAD system is used as long as required plugin is present. All interactions with CAD systems are generalized with a common interface so engineer can use any units. No conversions are applied.
- External methods allow using of user-defined algorithms and also can be a layer between the tool and external software.

<u>Optimization in DASY</u>

The genetic algorithm [3] (GA) is used to perform a multi-objective optimization. In genetic algorithm, a population of strings (called chromosomes or the genotype of the genome), which encode candidate solutions (called individuals, creatures, or phenotypes) to an optimization problem, evolves to-ward better solutions. In the current implementation of genetic algorithm solutions are represented as arrays of doubles. The evolution starts from a population of randomly generated individuals with respect to constraints and happens in generations. In each generation, the fitness of every individual in the population is evaluated, multiple individuals are stochastically selected from the current population (based on their fitness), and modified (recombined and possibly randomly mutated) to form a new population. The new population is then used in the next iteration of the algorithm. The algorithm terminates when a maximum number of generations has been produced. Genetic algorithm requires:

- a genetic representation of the solution domain,
- a fitness function to evaluate the solution domain.

A genetic representation of the solution is as an array of doubles. The fitness function is defined over the genetic representation and measures the quality of the represented solution. The fitness function is always problem dependent. In the current implementation, the fitness function returns values of the optimized parameters computed through the all dependencies in a model.

During each successive generation, a proportion of the existing population is selected to breed a new generation. Individual solutions are selected through a fitness-based process, where fitter solutions (as measured by a fitness function) are typically more likely to be selected. Certain selection methods rate the fitness of each solution and preferentially select the best solutions. In case of multi objective optimization, a Pareto selection is used. From two randomly selected solutions we take the one that is closer to Pareto front.

There is a set of best solutions or Pareto set. It is a set of compromises between several optimization goals. In each point there is a solution with the best combination of goals. It means that if we take any solution on Pareto front, then any other solution will have at least one parameter worse than the solution selected. Each solution on Pareto front is optimal and it is up to the engineer to decide which goal is more important and then choose an appropriate compromise. The final chosen optimum solution is saved in knowledge database.

OPTIMIZATION OF CRANKSHAFT

In order to perform optimization of crankshaft using DASY, the model of crankshaft should be connected to DASY model using Input & Output Files method. This method requires:

- Template of input file for ABAQUS, which is a Python script in this case (Fig. 9). Text marked with green will be substituted with input parameter values automatically.
- Rules defining the position of output parameter values in output file (after ABAQUS computation, Fig. 10).
- Commands to execute in order to start computation.

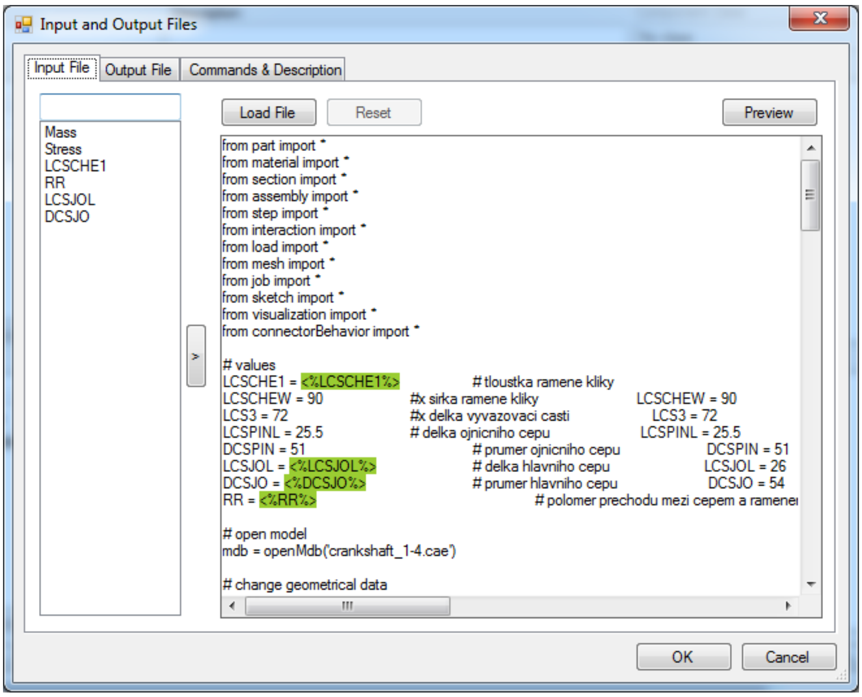

Fig. 9. Input file template.

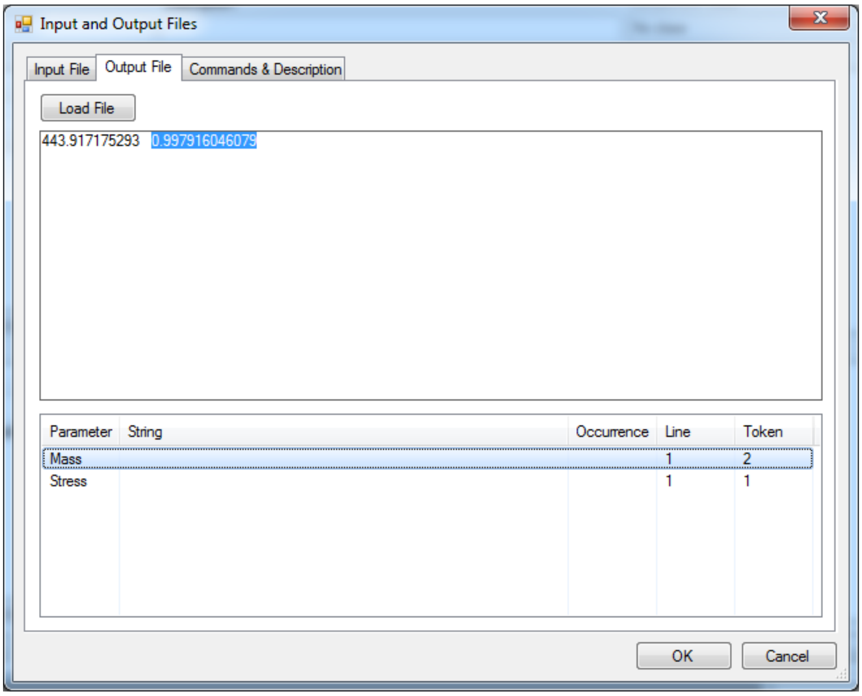

Fig. 10. Definition of output parameter values position in output file.

Parameters used for optimization are shown in Fig. 11. Following parameters were selected as variables:

- width of cheek, LCSCHE1;
- length of main journal (constraint in a way that LCSJOL + LCSCHE1 is constant), LCSJOL;
- diameter of main journal, DCSJO;
- fillet radius, RR.

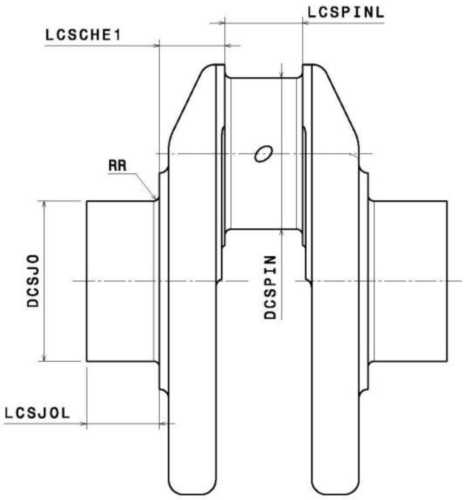

Fig. 11. Parameters for optimization.

The result of the optimization is a Pareto front shown in Fig. 12. Based on material strength and safety margins only one solutions from this set was selected. Final result of the optimization and limits for variables are shown in Table 1.

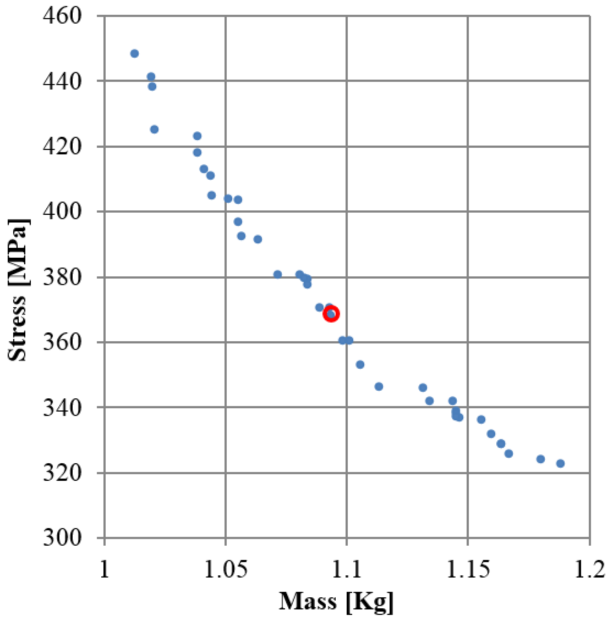

Fig. 12. Pareto set obtained as a result of optimization.

Table 1. Optimization result.

Parameter	Units	Original value	Limits	Optimal value
LCSCHE1	mm	21.5	18 – 23	19.4
LCSJOL	mm	26	47.5 – LCSCHE1	28.1
DCSJO	mm	54	50 – 56	52.5
RR	mm	2	1.5 – 2.5	2.5
Stress	MPa	412	minimize	460.1
Mass	kg	4.65	minimize	4.368

Because optimization was done using simplified model, a verification on a full model is required. Results of the verification are shown in Table 2.

Table 2. Results of verification.

	Stress [MPa]		Mass [kg]	
	before	after	bafore	after
simplified model	379.4	368	1.16	1.094
full model	412	460.1	4.65	4.368

As a result of this optimization 281.6 grams of material was saved for one crank. The value of maximal stress is below material strength limit. In this study the crankshaft was loaded only with a force acting on connecting rod journal. Real operating conditions are more complicated, so this study is a preliminary study only. Mass of the original crankshaft was 17.651 kg, mass of the optimized crankshaft is 16.571. Which means that 1080 grams of material was saved for the whole crankshaft.

CONCLUSIONS

Finite-element (FE) model of crankshaft was created in ABAQUS using geometry, measured on a real crankshaft. Simplified FE model of one crank was created using two planes of symmetry in order to reduce computational time. This allowed this simplified model to be used in the optimization loop. Preliminary optimization using DASY software was performed with two optimization goals: minimize mass and stress. The result of multi-objective optimization is a Pareto set, from which one solution was chosen based on material strength. Chosen solution was verified using full (not simplified) FE model to make sure that the stress values are not exceeding material strength limit. As a result of this preliminary study, 1080 grams of material were saved, which is a bit more than 6% of the original crankshaft mass.

REFERENCES

[1] Bogomolov S., Mikulec A., Macek, J. “Development of Design Assistance System and Its Appli-cation for Engine Concept Modeling”, SAE Technical Paper 2011-37-0030, 2011, doi: 10.4271/2011-37-0030

[2] Hejlsberg A., Torgersen M., Wiltamuth S., Golde P. “C# Programming Language, The (4th Edi-tion)”, 2010

[3] Banzhaf W., Nordin P., Keller R., Francone F. “Genetic Programming – An Introduction”, Morgan Kaufmann, San Francisco, CA, 1998

[4] Novotny T. “Analýza klikového hřídele vznětového, 4-válcového, řadového motoru”, Diploma Thesis, Czech Technical University in Prague, 2013

[5] http://msdn.microsoft.com

[6] "XML 1.0 Specification". W3.org.