Final design interpretation of the complex-shaped beam optimizations for compliant mechanisms

M. Sauter T. Kern P. Ermanni

Abstract—Compliant mechanisms are one-piece devices that combine the features of both structures and mechanisms. Genetic algorithms have been given increasing attention in the scientific community. Parsons and Canfield [14] explored genetic algorithms for the multi-criteria topology optimization of compliant structures. Saxena [15] presented a procedure to synthesize compliant mechanisms for a prescribed nonlinear output path. Lu and Kota [16] used a load path representation method to efficiently exclude the invalid topologies (disconnected structures) from the genetic algorithm solution space in order to avoid useless function evaluations. Zhou and Ting [17] introduced the spanning-tree theory for the topological synthesis of compliant structures. The spanning-tree theory is based on the graph theory where a graph consists of vertices and edges. A valid topology is regarded as a network connecting input, output, support, and intermediate nodes (vertices), which contains at least one spanning-tree among the introduced nodes. Akhtar et al. [18] combined the graph theory, genetic algorithms, and Bezier curves representation. The Bezier curves represent the topology of the structures. In a second step these curves are mapped into a continuum Finite Element Model (FEM).

The compliant gripper design is a commonly benchmarking problem that has been broadly studied [14], [19], [20], [21], [22]. The synthesis procedures can be typically divided into four steps [21]:

1. Problem specification (design domain and applied boundary conditions)
2. Design domain parametrization (design variables)
3. Topology optimization
4. Final design interpretation

This paper addresses the final design interpretation of the complex-shaped beam optimization of compliant grippers. The major challenge is to transfer the structural information into a manufacturable design without changing the optimized properties. Three different methods for final design interpretation are proposed and their efficiencies are discussed. The first two are CAD (Computer Aided Design) and the latter is Finite Element (FE) based.

The complex-shaped beam optimization was developed in an earlier work. The evolutionary optimization combines the
graph theory [23] with the curved, variable thickness beam element [24] which increase both numerical efficiency and solution space.

In Section II a brief overview of optimization setup is given including complex-shaped beam parametrization, topology representation and control routines which guarantee feasible structures. Section III explains the three different methods for final design interpretation. Section IV discusses the efficiency of the methods. Section V summarizes the results and gives an outlook on future research.

II. OPTIMIZATION SETUP

A. Complex-shaped beam element

The development of a beam element with high shape complexity, such as variable thickness and curved centerline shape, was motivated by simultaneously increasing the design flexibility and reducing the number of elements for a complex topology [24]. Additionally, this beam is able to place compliant regions within it.

Fig. 1 illustrates the shape complexity and how it depends on the parameters used in this work.

Fig. 1. Curved, variable-thickness beam

Many compliant mechanism design problems require finding a best topology as well as an optimized shape. Often the two aspects are separated. A first method is used for finding the topology and a second-step method refines the result by obtaining the best shape solution for the fixed topology. The shape flexibility of this element offers the possibility to combine the topology and thickness distribution optimization within one computational process.

We use Castigliano’s theorem [25] for connecting the set of beam parameters with the beam’s structural properties because it provides a good compromise between accuracy and numerical costs for linear elasticity problems [24], [26].

In the following the parametrization of the complex-shaped beam is presented.

For the curved centerline the Hermite curve representation

\[
x(\sigma) = a_0 + a_1 \sigma + a_2 \sigma^2 + a_3 \sigma^3 \\
y(\sigma) = b_0 + b_1 \sigma + b_2 \sigma^2 + b_3 \sigma^3
\]  

(1)

is chosen, where \(\sigma\) is in the range from -1 to 1.

The coefficients of these two equations depend on the coordinates of the nodes with which the element is connected, the chosen angle at the nodes, and the factor L.

\(L\) influences the line length of the curved centerlines (See Fig. 2).

\[
\frac{dx}{d\sigma}(\sigma = -1) = a_1 - 2a_2 + 3a_3 = L \cos(\beta_1), \\
\frac{dy}{d\sigma}(\sigma = -1) = b_1 - 2b_2 + 3b_3 = L \sin(\beta_1), \\
\frac{dy}{d\sigma}(\sigma = -1) = \frac{dy(\sigma = -1)}{d\sigma} = \tan(\beta_1). \\
\]

(2)

For the thickness distribution two cubic thickness functions are used

\[
T_1(\sigma) = t_0 + t_1 \sigma + t_2 \sigma^2 + t_3 \sigma^3, \\
T_2(\sigma) = t_4 + t_5 \sigma + t_6 \sigma^2 + t_7 \sigma^3
\]

(3)

where the first \(T_1\) goes from \(\sigma=1\) to 0 and the second \(T_2\) from \(\sigma=0\) to 1. There are eight variables which are defined by thickness values at the locations along the beam centerline as Fig. 1 illustrates and by the restrictions that the slope at \(\sigma=1\) and \(\sigma=1\) must be zero

\[
\frac{dT_1}{d\sigma}(1) = t_5 + 2t_6 + 3t_7 = 0, \\
\frac{dT_2}{d\sigma}(0) = t_4
\]

(4)

and that the two function must be steady at \(\sigma=0\)

\[
T_1(0) = t_0 = T_2(0) = t_4
\]

(5)

The parameter set for describing one complex shaped beam element includes the values of

- 4 end-point coordinates
- 2 centerline run-out angles at the end points,
- 1 factor \(L\), and
- 4 thickness points along the centerline.

The choice of material, if desired for multi-material optimization, adds a twelfth parameter to the set. The shape complexity requires six more parameters than would be necessary if conventional beam elements were used.

B. Graph topology representation

The complete topology is represented by a graph, wherein each vertex corresponds to a node and each beam is described by an edge (Fig. 3). In other words, instead of holding the information of the beam structure in a conventional 1-dimensional genotype, the graph itself is considered as the
genotype. The operators are directly applied on the graph genotype.

A graph is an abstract mathematical model; it is an ordered pair \((V, E)\), where \(V\) is a finite set called vertex set and \(E\) is a binary relation on \(V\) called edge set. Elements of \(V\) are called vertices, elements of \(E\) edges.

### TABLE I

<table>
<thead>
<tr>
<th>edge</th>
<th>(l_1)</th>
<th>(l_2)</th>
<th>(\beta_1)</th>
<th>(\beta_2)</th>
<th>(T_1)</th>
<th>(T_2)</th>
<th>(T_3)</th>
<th>(T_4)</th>
<th>(L)</th>
<th>status</th>
</tr>
</thead>
<tbody>
<tr>
<td>(e_0)</td>
<td>0</td>
<td>3</td>
<td>0.0</td>
<td>4.3</td>
<td>1.0</td>
<td>7.7</td>
<td>10.7</td>
<td>8.6</td>
<td>2.1</td>
<td>2</td>
</tr>
<tr>
<td>(e_1)</td>
<td>2</td>
<td>3</td>
<td>-1.2</td>
<td>3.2</td>
<td>1.0</td>
<td>9.9</td>
<td>15.8</td>
<td>9.2</td>
<td>2.0</td>
<td>2</td>
</tr>
<tr>
<td>(e_2)</td>
<td>3</td>
<td>1</td>
<td>0.4</td>
<td>0.4</td>
<td>1.0</td>
<td>5.4</td>
<td>8.5</td>
<td>8.5</td>
<td>1.0</td>
<td>2</td>
</tr>
</tbody>
</table>

### TABLE II

<table>
<thead>
<tr>
<th>vertex</th>
<th>(l)</th>
<th>(x)</th>
<th>(y)</th>
<th>moveable</th>
</tr>
</thead>
<tbody>
<tr>
<td>(v_0)</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td>false</td>
</tr>
<tr>
<td>(v_1)</td>
<td>1</td>
<td>100.0</td>
<td>0.0</td>
<td>false</td>
</tr>
<tr>
<td>(v_2)</td>
<td>2</td>
<td>0.0</td>
<td>50.0</td>
<td>false</td>
</tr>
<tr>
<td>(v_3)</td>
<td>3</td>
<td>49.2</td>
<td>49.9</td>
<td>true</td>
</tr>
</tbody>
</table>

According to the Section II-A seven parameters, e.g. angles and thicknesses, are attached to the edges and two to the vertices of the graph as properties, respectively.

In addition two parameters are introduced.

- a boolean moveable defines whether or not node coordinates may change.
- an unsigned number attached to the edge elements describes the status of the edge/beam.

Three settings are possible:

0 endpoints and shape (the four thickness parameters) are fixed
1 only endpoints are fixed
2 endpoints and shape can be modified

Consequently, edges with status 0 or 1 must not be removed or added from/to the graph and from/to the structure during the optimization.

### C. Control routines

The control routines guarantee that only legal design solutions are evaluated. Under illegal design solutions we understand that the parts of the domain boundary, where non-zero forces and/or displacements are specified, are not connected, so that the mechanism is not realized and/or the stiffness matrix remains singular. Also, design solutions not complying with mechanical modelling assumptions must be rejected. We distinguish two types of control routines.

The first one regards the FEM analysis and checks whether the topology connects the above mentioned parts of the boundary and whether all beam elements are sufficiently long with respect to their thickness to comply with beam theory assumptions. The second one looks for crossing of beams and includes either:

a) control routine for crossing centerlines or
b) control routine for overlapping zones

Control routine a) has only to check the crossing of the centerline of each beam with the centerline of other beams (Fig. 4). Since overlapping almost always occurs (Fig. 5) if two or more beams are connected by the same node, the routine b) divides the beams into restricted and non-restricted overlapping zones (Fig. 6). The overlapping zones routine detects overlapping of restricted zones or overlapping of symmetry lines if at least one is defined.

Fig. 4 and Fig. 5 visualize the differences between the routines using the same example.

In contrast to crossing centerlines routine the overlapping zones routine does punish this configuration.

Avoiding overlapping zones contributes to finding manufacturable designs. Nevertheless it narrows the design space and makes it more difficult to find a suitable topology for the mechanism which decreases the convergence. By experience it seems most efficient to use the crossing centerline routine for finding the best topology and to use the overlapping elements routine for refinements.

### D. Optimization setup

Fig. 7 displays optimization setup. A straight beam at the output is prescribed and four spring are added in order to exert a structural resistance. The problem is geometrically symmetric which reduces the design domain \(\Omega\) by a half.
The energy in the system is limited and first the optimization maximizes the transmitted energy $w_{transmitted}$

$$w_{transmitted} = w_{out,1} + w_{out,2},$$

where $w_{out,1}$ the output energy at the first spring and $w_{out,2}$ the output energy at the second spring.

Since the lower output energy at either the first or second spring is maximized and since the algorithm seeks to exploit the whole amount of transmitted energy, optimization leads to a result where the output energies at both springs are the same and where parallel deflections are achieved.

The result of the optimization shown in Fig. 8 supports this explanation.

The FEM evaluation done by our in-house developed tool FELyX [27] assumes exact parallel deflections (Table III).

<table>
<thead>
<tr>
<th>program</th>
<th>$w_{out}$</th>
<th>$v_1$</th>
<th>$v_2$</th>
<th>diff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parallel gripper FELyX</td>
<td>-0.0336</td>
<td>-0.08195</td>
<td>-0.08195</td>
<td>0.0%</td>
</tr>
</tbody>
</table>

### III. Three Methods for Final Design Interpretation

Generally our postprocessing has three objectives:

1) to check of the fulfilling of secondary constraints which were neglected in order not to slow down the optimization
2) to analyse of the structural behavior in terms of variation of the loads
3) to transfer of the optimized structures into a manufacturable design (final design interpretation)
In the following three different methods for the final design interpretation are presented and their accuracies of the resulting final designs are compared with the solution of the optimization. Two of them are CAD based. The CAD based procedure can be described as follows; a MATLAB \[28\] routine generates a Visual Basic Script including the structural information of the optimized structure. This script can be directly loaded into the CAD program CATIA V16 \[29\]. Thereby the user is free to manually modelling the structure by connecting the points previously generated in Catia or to establish the CAD Model automatically. Then the model can be exported into ANSYS \[30\], a commercial FEA software, for further analysis.

The third final design interpretation procedure generates plane elements by using a MATLAB interface again which can be directly loaded into ANSYS for postprocessing.

In the course of this work the interpretation of the parallel gripper optimization displayed in Fig. 8 is considered.

A. Manual CAD based final design interpretation

As mentioned before the manual CAD based final design interpretation procedure generates a set of nodes which describe the contour of the 2-dimensional structures. The designer can manually connect these nodes using splines. Because of the same nature of the splines and of the parametrization of complex-shaped beam structures the CAD model matches very good with optimized structure even if less contour nodes are employed.

The manual establishing allows to modify the structures by shifting or skipping nodes incorporating the designer's experiences.

Fig. 9 shows the contour nodes manually connected by splines.

The incorporation of the designer's knowledge seems to be advantageous due to multiple interpretation possibilities of the transition zone of the beams next to the nodes and due to the limited solution space of the cubic polynomial thickness parametrization.

Fig. 10 displays the manually generated CAD based parallel gripper.

B. Automated CAD based final design interpretation

In contrast to the previous method this method generated the CAD model automatically. The method uses the same approach as the manual method and connects the contour nodes of each beam using splines. The greatest challenge is to overcome the gap between the beams (see Fig. 11).

The method adds a chamfer at the ends of the beams which overlap with each other and close the undesired gaps. An additional logical operation unites the beams. The added chamfer can be manipulated by the tension factor T; T=0.01 for an approximate straight line and T=2.0 for an approximate semi-cycle. By experience the use of tension factor T=1.0 has often given good results.

Fig. 12 presents the automated CAD Model.
Fig. 12. Automated CAD based parallel gripper

C. Element based final design interpretation

The development of the element based final design interpretation was motivated by avoiding the CAD intermediate step and by giving the opportunity of simultaneous optimization of the transition zone of the beams in future. The procedure can be divided into three steps.

1) Mapping each beam into a number of 4-nodal-elements
2) Deleting the overlapping elements
3) Connecting the beams using triangular elements.

Fig. 13 presents the element based final design interpretation of the parallel gripper.

In contrast to the optimization, where the computation incorporates six beam elements for the semi-symmetric gripper model, the element based final design interpretation has to evaluate more then 60 plane elements, which obviously affects the computing time. However, the computing time is not critical for our final design interpretation.

IV. MAIN RESULTS

In this section the three previously presented methods are compared with each other and their efficiencies are discussed. Thereon the result of an optimization using the enhanced control routine for overlapping zones is presented. The routine seeks to influence the optimization in order to minimize the overlap and thereby more efficient final design interpretation can be obtained.

As mentioned before the main challenge is to transfer the optimized structure into a manufacturable design without loosing the structural properties. As properties the optimized energy at the output (Eq. 6) and the difference of the displacements, which describes the parallelism of the gripper output, are considered. Both are normalized; the output energy by the output energy of the optimized structures and the difference of displacements by the average value of the displacements.

Tables IV and V summarize the results of the three methods using geometrically linear and nonlinear computation, respectively.

<table>
<thead>
<tr>
<th>TABLE IV</th>
<th>COMPARISON OF THE METHODS; USING GEOMETRICALLY LINEAR COMPUTATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>method</td>
<td>program</td>
</tr>
<tr>
<td>optimization</td>
<td>FELsX</td>
</tr>
<tr>
<td>manual CAD</td>
<td>ANSYS</td>
</tr>
<tr>
<td>automated CAD</td>
<td>ANSYS</td>
</tr>
<tr>
<td>element based</td>
<td>ANSYS</td>
</tr>
</tbody>
</table>

The difference between the linear and nonlinear computation is for this problem very small and therefore optimization using linear computation was legitimated.

<table>
<thead>
<tr>
<th>TABLE V</th>
<th>COMPARISON OF THE METHODS; USING GEOMETRICALLY NONLINEAR COMPUTATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>method</td>
<td>program</td>
</tr>
<tr>
<td>manual CAD</td>
<td>ANSYS</td>
</tr>
<tr>
<td>automated CAD</td>
<td>ANSYS</td>
</tr>
<tr>
<td>element based</td>
<td>ANSYS</td>
</tr>
</tbody>
</table>

The loss of output energy of the CAD methods is 3.3% and 3.1%, respectively, and the difference of the displacement is less then 1.2%. This loss can be explained by the increased stiffness next to the nodes and the reduced free length of the
The use of complex-shaped beam finite elements, instead of two-dimensional elements, accelerates the evolutionary algorithm optimization process of two-dimensional compliant-mechanism design but leaves the problem that overlapping of geometry may occur in regions close to nodes, which is addressed here. Three methods for solving the problem, which we regard as final design interpretation, have been proposed. The first one generates points contours of the complex-shaped beams and these are manually fitted with splines, smoothing out the regions where the beams connect. The second uses a Visual Basic Script to perform the fitting automatically where the logical operation for gaps and overlaps has here been explained. The third approximates the beam contour geometrically with two-dimensional finite elements. The final design interpretation methods reduce the mechanism efficiency values found by the beam model. The reductions of the three design interpretation methods have been investigated. A maximum output energy loss of about eight percent is incurred by the element-based method. The other two methods produce models with smaller efficiency losses at around three percent. However, the element-based method offers the advantage of being independent of any CAD software and its by-product, the two-dimensional solid model, can immediately be used for a refining optimization step. Finally the best choice of the three methods depends on the user’s requirement.

V. CONCLUSIONS

The use of complex-shaped beam finite elements compared to the beam representation, whereas the beams are connected by nodes.

The element based final design interpretation reduces the output energy by 8.1% and also the difference of the displacements is about three times higher than for the CAD methods. The reason lies in the relative rough approximation of the structures. Nevertheless it is the fastest final interpretation method and independent of each CAD software.

The aim of using the control routine for overlapping zones is to manipulate the optimization in order to increase the accuracy of the later final design interpretation. But each manipulation, which is an additional constrain in our case, limits the solution space. The optimization using the control routine for overlapping zones results in 98.2% of the energy of the previous optimization.

<table>
<thead>
<tr>
<th>method</th>
<th>program</th>
<th>(w_{\text{out}}) [mm]</th>
<th>(v_1) [mm]</th>
<th>(v_2) [mm]</th>
<th>(\text{diff.})</th>
</tr>
</thead>
<tbody>
<tr>
<td>optimization</td>
<td>FELyX</td>
<td>100.0%</td>
<td>-0.08122</td>
<td>-0.08139</td>
<td>0.2%</td>
</tr>
<tr>
<td>automated CAD</td>
<td>ANSYS</td>
<td>97.2%</td>
<td>-0.08009</td>
<td>-0.08101</td>
<td>1.1%</td>
</tr>
</tbody>
</table>

The accuracy of the automated CAD based method is slightly better (97.2% see Table VI). But finally the output energy of the final design is lower (97.2% + 98.2% = 95.5%) than in the previous case. Even though this approach increases the accuracy, an improved design in terms of the output energy could not be achieved.

ACKNOWLEDGMENT

The authors would like to thank Dr. Gerald Kress for his help and his fruitful advice.

REFERENCES


