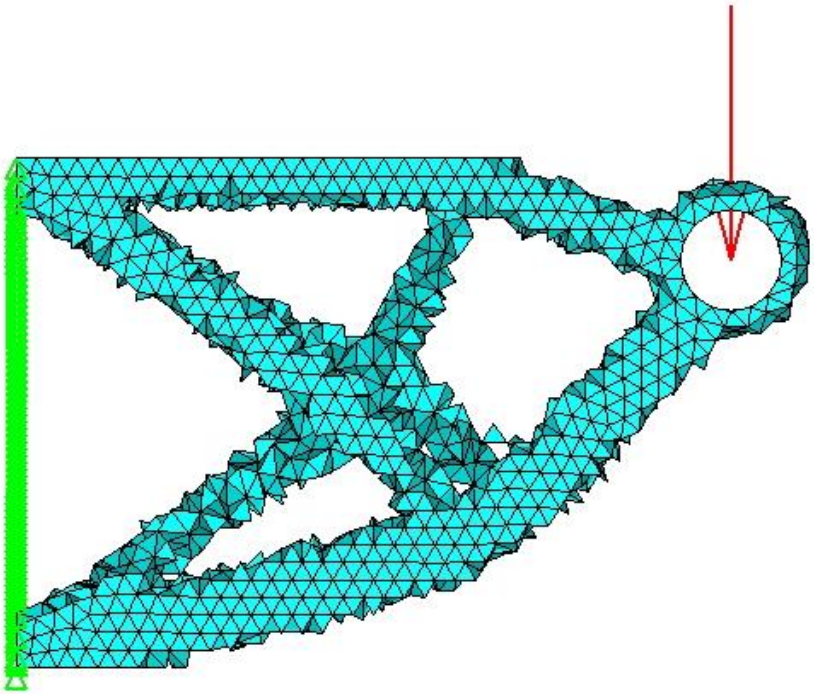
OptiStruct® Tutorial

Version 11.0

Topology Optimization using Altair® HyperWorks® 11.0



Eduardo Moreno Ruiz

Copyright ©2013 by Eduardo Moreno Ruiz. All rights reserved.

No part of this document may be reproduced in any form or by any means without written permission from the author. This document may not be transferred to any other party.

The author has used his best efforts in preparing this document. The efforts include the testing of the tutorial to determine its effectiveness. However, the author makes no warranty of any kind, expressed or implied, with regard to the material contained in this document. The author shall not be liable in any event for incidental or consequential damages in connection with the use of the material contained herein.

Trademarks:

Altair® HyperWorks®, HyperMesh®, RADIOSS®, HyperView® and OptiStruct® and any and all Altair Engineering Inc. product and service names are registered trademarks of Altair Engineering Inc. or its subsidiaries located in the United States or other countries. All other trademarks or registered trademarks are the property of their respective owners.

to Marco

for giving me the first lesson

to my parents

for giving me freedom

to Dani

for being the best brother

and to Jésica

for being my cacharrito

Contents

Introduction **1**

Example: Beam **2**

Design Space (CAD) 3

Importing to HyperMesh 3

Material 7

Mesh 9

Properties 10

Components 13

Boundary Conditions 15

Rigid Elements 17

Loads 18

Loadsteps 19

Design Variable 20

Assign Properties 21

Responses 22

Constraints and Objective 24

OptiStruct 27

HyperView 28

Postprocessor 29

CAD 32

Appendix A: FEA tips and tricks **34**

Appendix B: Maximum number of iterations **35**

Appendix C: Symmetry and Extrusion 36

Symmetry 36

Extrusion 39

Appendix D: Bearing Load 41

Appendix E: Contact Surfaces 42

Appendix F: Pivot 49

a) Rigid Elements 49

b) Contact Surfaces 49

Appendix G: OptiStruct Steps 51

Introduction

What is OptiStruct®?

OptiStruct® is the Topology Optimization module from the CAE software Altair® HyperWorks®. Other important modules of the software are:

- HyperMesh® (Preprocessor)
- RADIOSS® (Solver)
- HyperView® (Postprocesor)

What is Topology Optimization?

Topology Optimization is a type of structural optimization whose goal is to determine the location and shape of the holes of a structure as well as the connectivity of the domain.

The objective is to minimize a function taking into account some constraints. For example, the objective may be to minimize the compliance (i.e. maximize the stiffness) while a constraint is applied on the volume. By doing so, the volume of the structure will be reduced while maintaining the maximum level of stiffness possible.

That allows the designer to create very lightweight components.

Example: Beam

In the following example we are going to perform a Topology Optimization. The component is a steel beam welded on the left side and loaded on a hole on the right side. The load is applied by means of a bolt and therefore there will be a bearing load distribution on the hole.

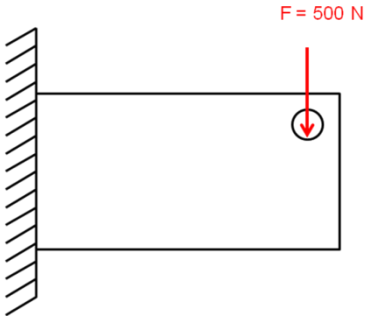


Figure 1 Beam model

The beam dimensions are showed in Figure 2.

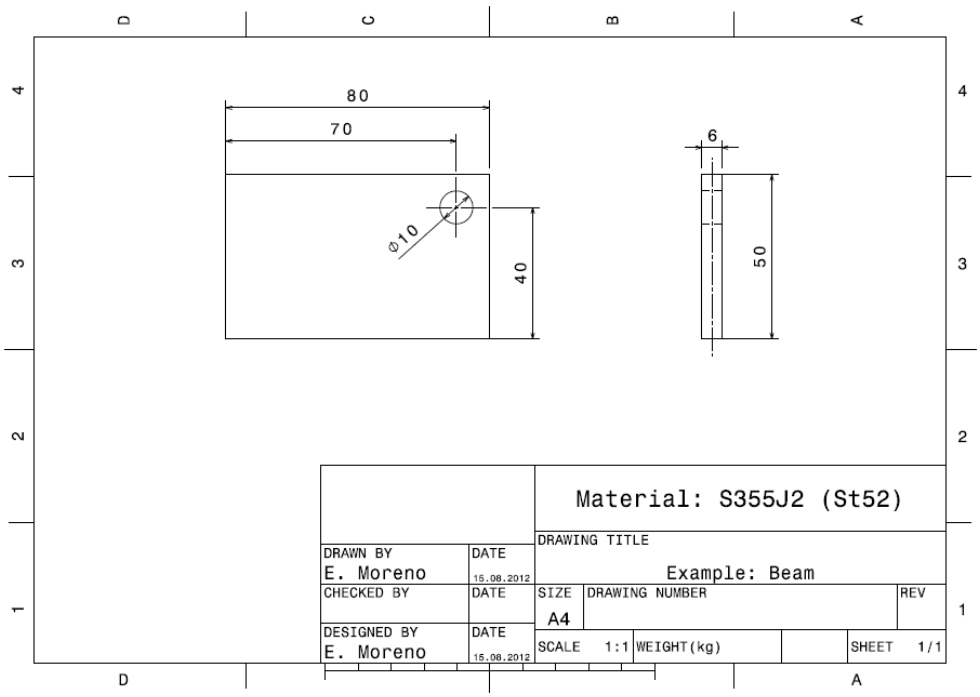


Figure 2 Beam drawing

Design Space (CAD)

The first step is to create a *Design Space* in CAD (for this tutorial CATIA® V5 will be used, but you can use any other CAD Software). This model should include the volume to be optimized and the necessary holes.

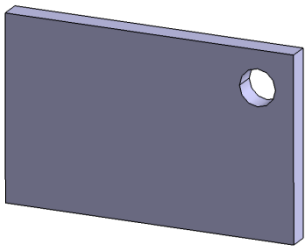


Figure 3 Design Space

Save the CAD as an IGS file.

Importing to HyperMesh

The next step is to import the model to HyperMesh. Start HyperMesh and select OptiStruct on the User Profiles' window:

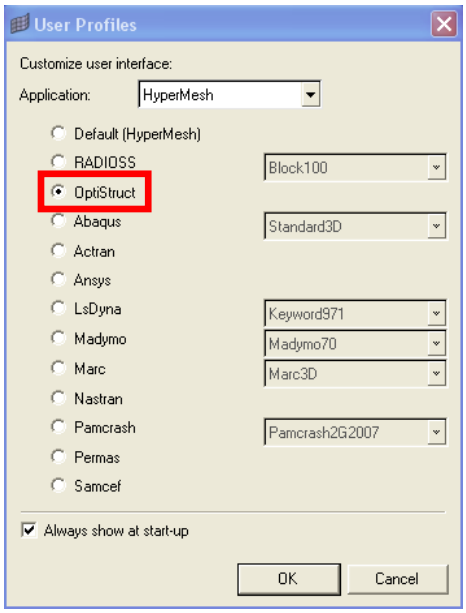


Figure 4 User Profiles

First we have to import the geometry into HyperMesh (**File/Import/Geometry**).

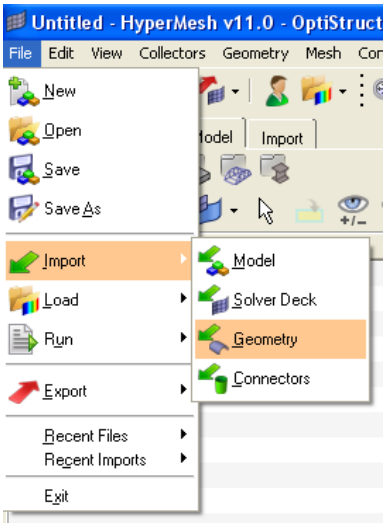


Figure 5 Import Geometry

Choose the IGS file and click **Import**.

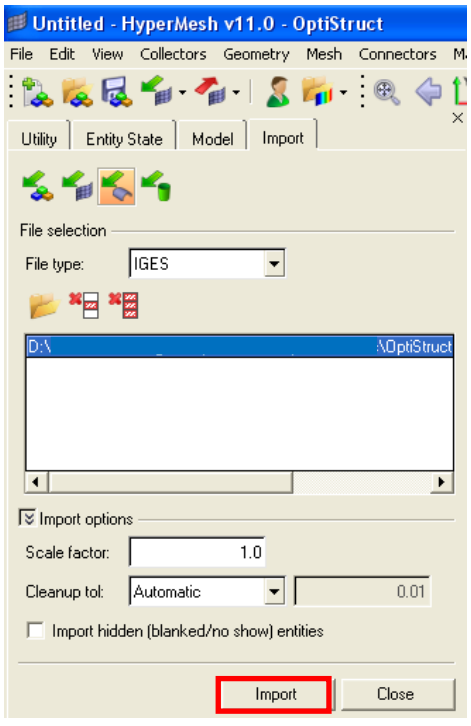


Figure 6 Import

Click **Model** and call the new component “Beam” (click with the right button of the mouse over the name and select **Rename**).

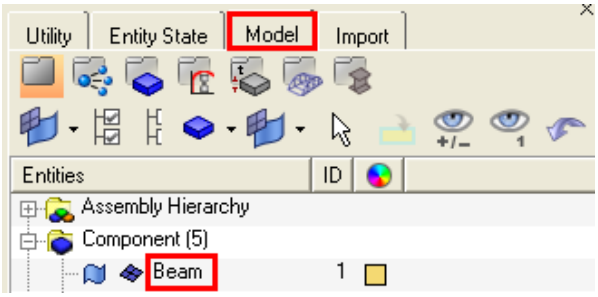


Figure 7 Rename the component

By pressing CNTRL you can move the model with the mouse:

- **CNTRL+left**: rotate
- **CNTRL+scroll**: zoom
- **CNTRL+right**: translate

Save the model (**File/Save** or).

Click the button **Shaded Geometry and Surface Edges** to visualize the faces of the model.

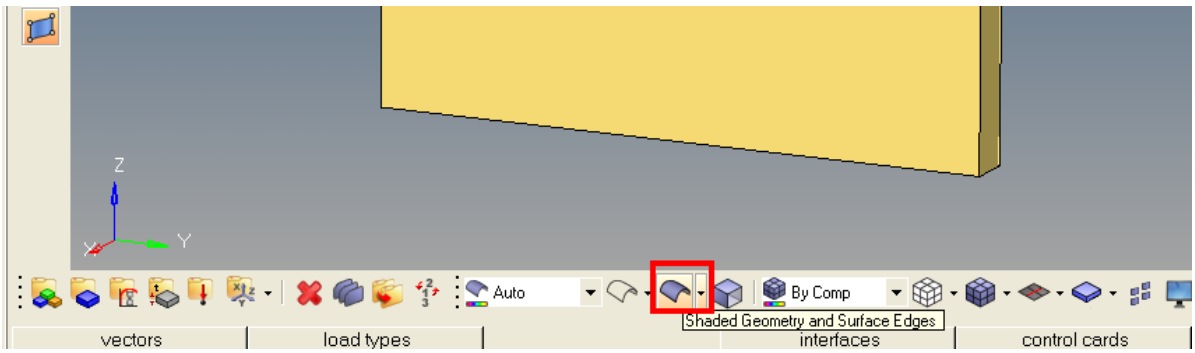


Figure 8 Shaded geometry and Surface Edges

Now we have to create the solid (we only have the geometry). Click **Geom/solids**.

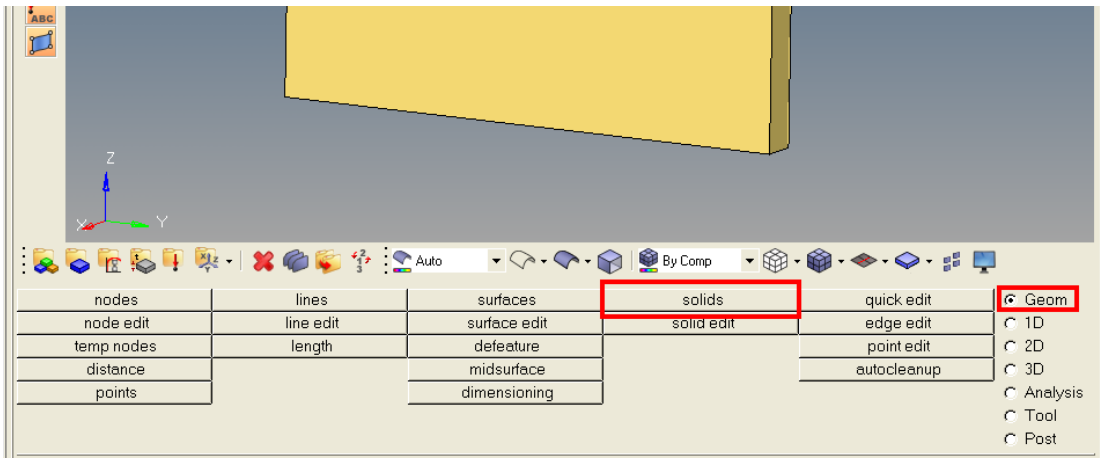


Figure 9 Solids

Then click **Bounding Surfaces**. Select the model (it has to turn white) and click **create**. Then click **return**. Save the model.

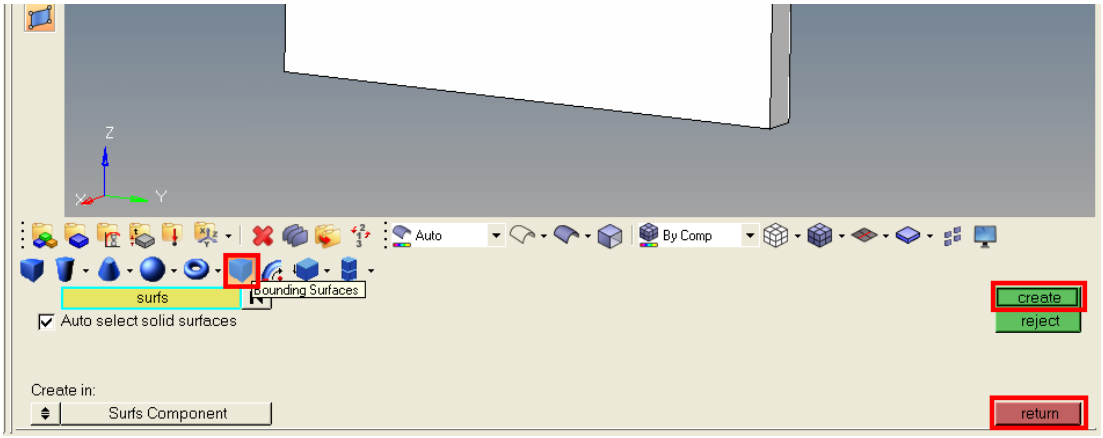


Figure 10 Creating the solid