NOVÉ TECHNOLOGIE
VÝZKUMNÉ CENTRUM
ZÁPADOČESKÉ
UNIVERZITY
V PLZNI

ODBOR MODELOVÁNÍ HETEROGENNÍCH MATERIÁLŮ A BIOMECHANICKÝCH SYSTÉMŮ

AUTORIZOVANÝ SOFTWARE

SCALE2DIM2010

SOFTWARE FOR SCALING SEGMENTS

Autor: *Ing. Hana Čechová, Ph.D.*
Ing. Luděk Hynčík, Ph.D.

Číslo projektu: *N*

Číslo výsledku: *NTC-SW-01-10*

Odpovědný pracovník: *Ing. Hana Čechová, Ph.D.*

Vedoucí odboru: *doc. Dr. Ing. Eduard Rohan*

Ředitel centra: *doc. Dr. RNDr. Miroslav Holeček*

PLZEŇ, KVĚTEN 2010

Jazyk výsledku: *ENG*

Hlavní obor: *JR*

Uplatněn: *ANO*

Název výsledku česky:

Scale2Dim - Software pro škálování segmentů

Název výsledku anglicky:

Scale2Dim - Software for scaling segments

Abstrakt k výsledku česky:

Scale2Dim - verze 2010 - umožňuje přenesení geometrie původního modelu člověka nebo figuríny na jiný cílový model člověka nebo figuríny při zachování rozměrů cílového modelu. Software pracuje s modely na bázi tuhých těles.

Abstrakt k výsledku anglicky:

Scale2Dim - version 2010 - enables transformation of original human or dummy model geometry to a target human or dummy model. Target model proportions are saved. The software should be used on rigid body based models.

Klíčová slova česky:

tuhé těleso;model člověka;model figuríny;škálování

Klíčová slova anglicky:

rigid body;human model; dummy model;scaling

Vlastník výsledku:	*Západočeská univerzita v Plzni*
IČ vlastníka výsledku:	*49777513*
Stát:	*Česká republika*
Lokalizace:	*http://www.zcu.cz/ntc/vysledky/sw/NTC-SW-01-10.html*
Licence:	*ANO*
Licenční poplatek:	*NE*
Ekonomické parametry:	*Urychlení biomechanických analýz a s tím související snížení nákladů*
Technické parametry:	*Luděk Hynčík, Západočeská univerzita v Plzni, Nové technologie - Výzkumné centrum v západočeském regionu, Univerzitní 8, 306 14 Plzeň, 377634709, hyncik@ntc.zcu.cz*

SOFTWARE FOR SCALING RIGID BODY BASED MODELS

The Scale2Dim software serves for scaling of geometry of the original model to the target model. It is written for models implemented in the PAM-system specially. The software usage is supposed to transform the Robby model geometry to another model hence it is based on the Robby model numbering and structure. It is possible to use arbitrary original model however it is necessary to keep the rigid body numbering and structure.

Methods

The both original and target models are divided into 17 segments. Each segment is associated to one number (segment ID) as defined in Table 1 and each segment includes one or more rigid bodies, see also Figure 1. If there is any nonstructural node (does not belong to any rigid body) it is assigned to a segment with ID 0. These nodes are not further included in the scaling process.

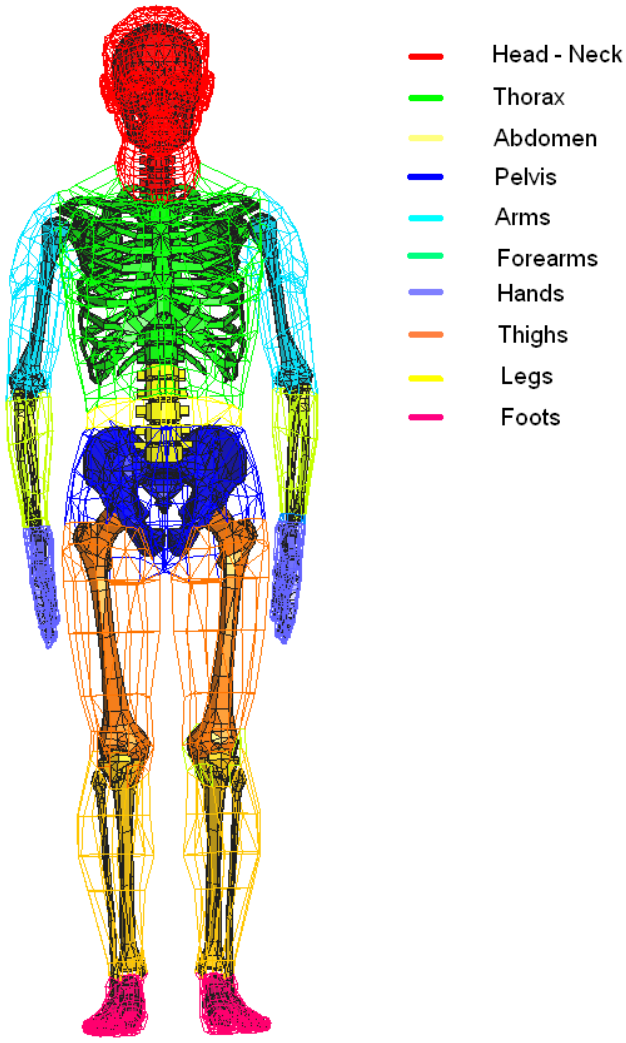


Figure 1: Segmentation of the Robby model.

Segment name	Segment ID	Rigid body ID
Head	1	100
Neck	2	200 – 208
Thorax	3	300 – 319
Abdomen	4	400 – 406
Pelvis	5	410
Left thigh	6	700
Left leg	7	730
Left foot	8	760
Right thigh	9	800
Right leg	10	830
Right foot	11	860
Left arm	12	500
Left forearm	13	530, 531
Left hand	14	560
Right arm	15	600
Right forearm	16	630, 631
Right hand	17	660

Table 1: Original model segmentation.

Each segment (both the original and the target) is defined by two nodes, its origin (J1) and end (J2), see Figure 2. Using these nodes the local coordinate system of the segment is defined and a the length parameter (abs(**v**)) for scaling is determined.

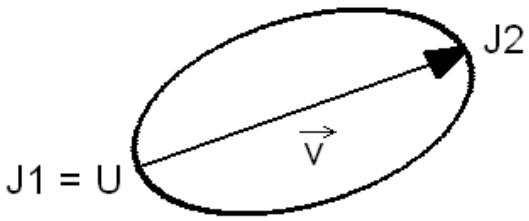


Figure 2: Schematic representation of a segment and its characteristic nodes.

The coordinates of all nodes belonging to a particular segment of the original model are multiplied by scaling coefficients and transformed to a proper coordinate system of the target model. Scaling coefficients are determined as the rate of the original and target segment size. Except the thoracic and pelvic segment it is supposed the uniform scaling in all directions hence the scaling coefficients are identical for all three main local directions. For the thoracic and pelvic segments the transversal and vertical coefficients are computed. New nodes representing the original geometry scaled to the target size are exported in the PAM-Crash specific file format.

Implementation

Installation and run

The source code is implemented in the Python scripting language. To run Scale2Dim, the user should install the Python language (for example Python 2.6.2, accessible at www.python.org). The installation is available for any kind of platform.

The software can be started up through Python application (e.g. Pylab) or using the standard command line:

```
>> Scale2Dim_gui.py
```

Then the user is appealed to choose the original *pc* file, original *out* file and target *pc* file and further the files containing nodes for segments determination.

Structure of input files

The code requires five input files described below. Three files are related to the original model (**.pc*, **.out*, **.txt*) and two files are related to the target model (**.pc*, **.txt*).

**.pc* – the model input file. This file has to contain coordinates of all structural nodes. It is required for both the original and the target model.

**.out* – the model output file. This file is generated as the output file of the PAM-Crash solver. It is used for allocation of nodes to proper rigid bodies. It has to contain all rigid bodies of the original model. This file is required only for the original model.

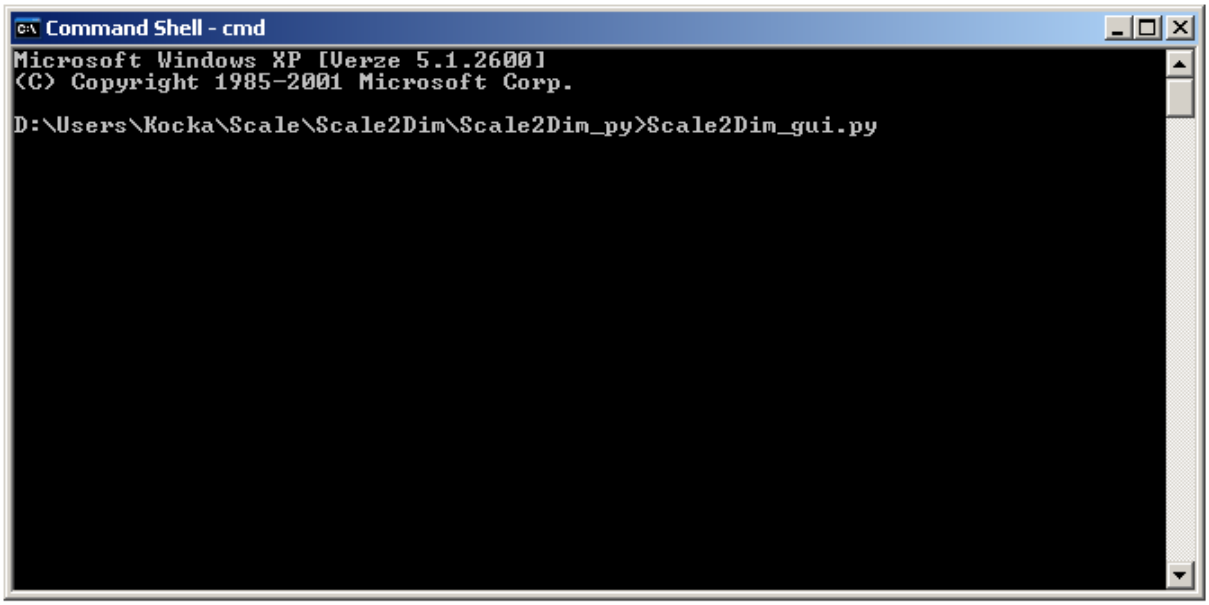
**.txt* – user defined file. This file contains ID numbers of nodes determining the origin and the end of each segment. The nodal IDs are ordered in two columns so that each row corresponds to the particular segment (the row number indicate the segment number). The file must have 17 rows indicating 17 segments as defined in Table 1. For segment determination it is recommended to choose joint nodes. This file is required for both the original and the target model.

Remark: the target **.pc* file has to contain coordinates of nodes listed in target **.txt* file particularly.

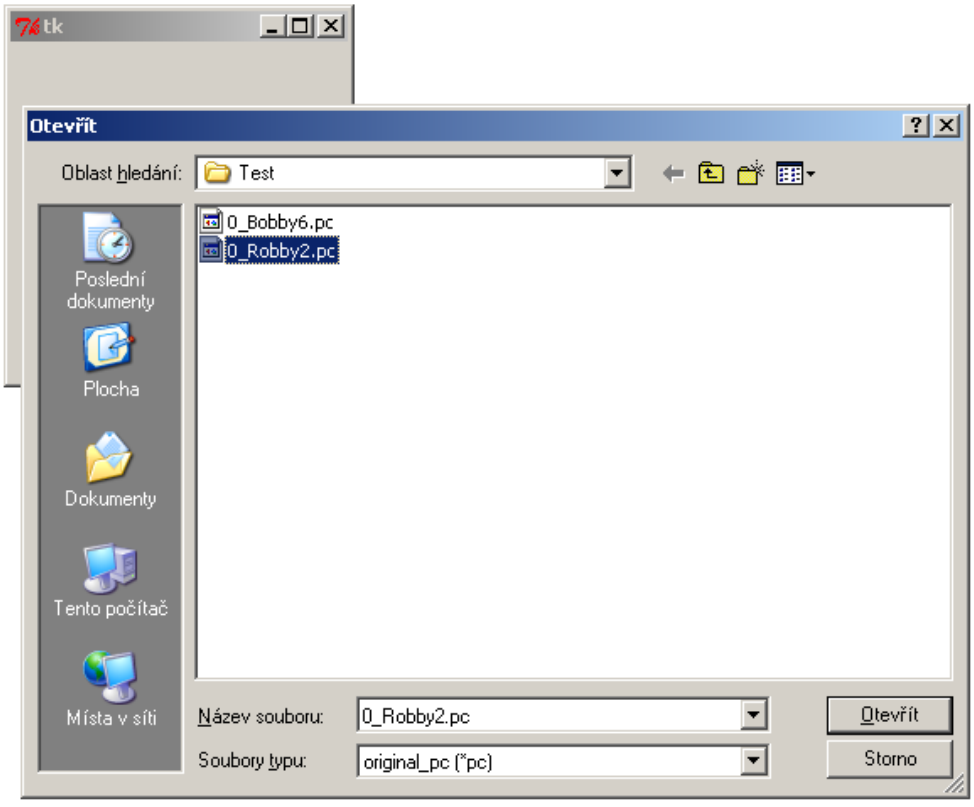
Structure of output file

ScaledNodes.inc - the software output file. It contains all nodal IDs and scaled coordinates exported in the PAM-Crash input format.

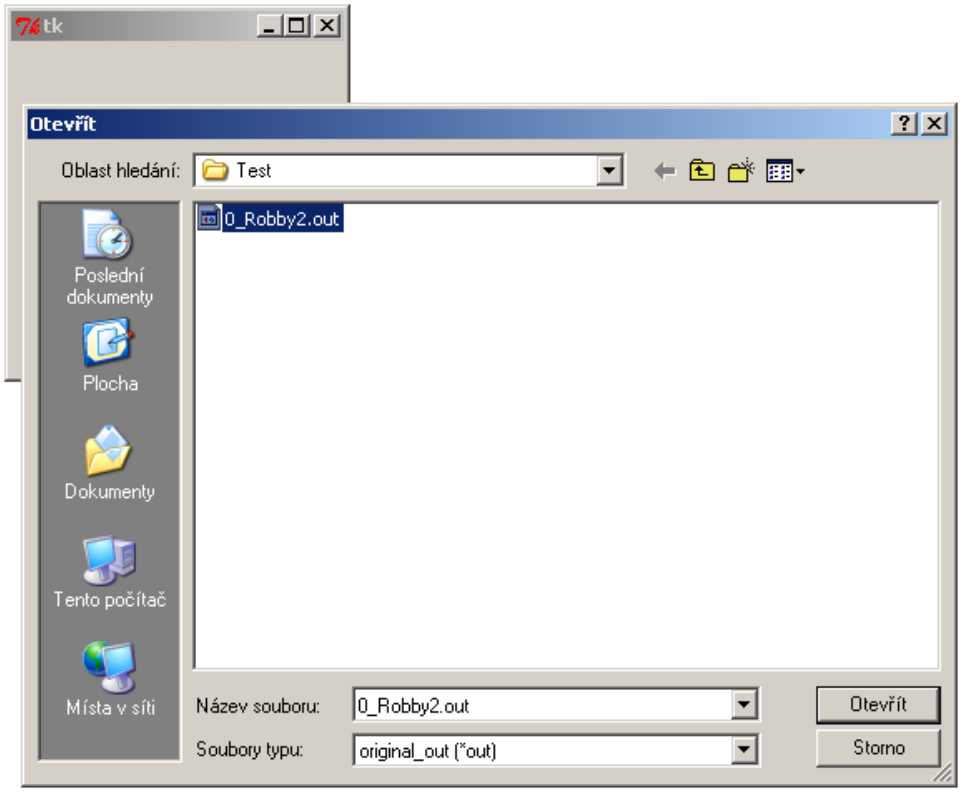
Example



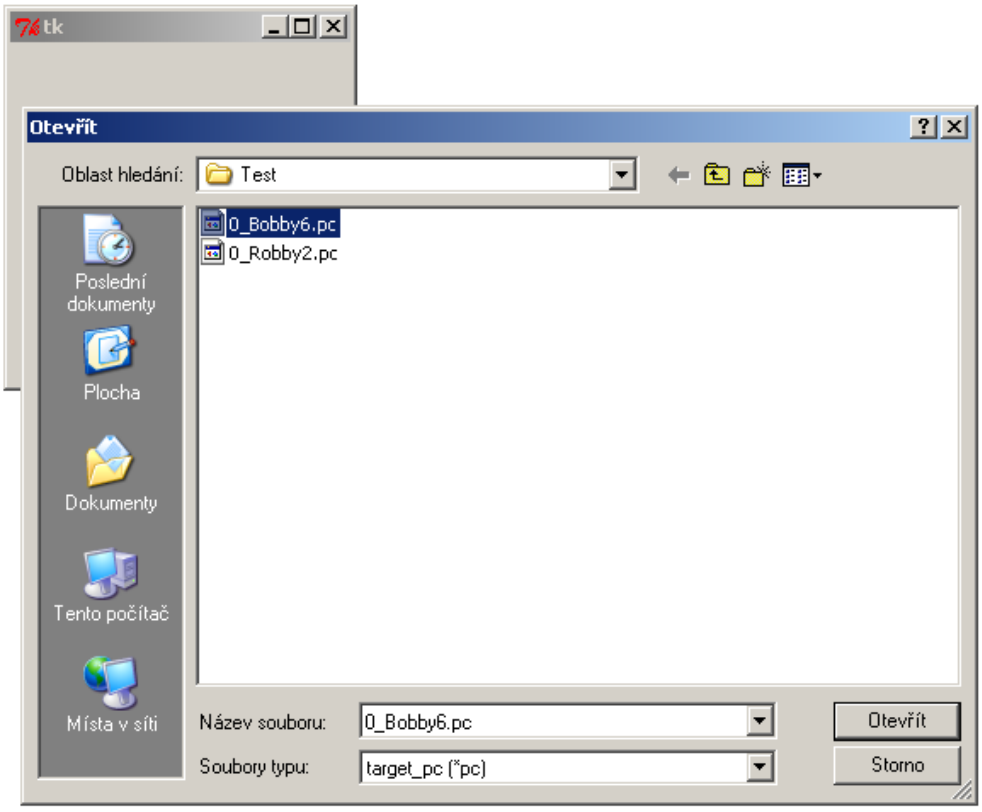
Step 1: Run the software from the command line



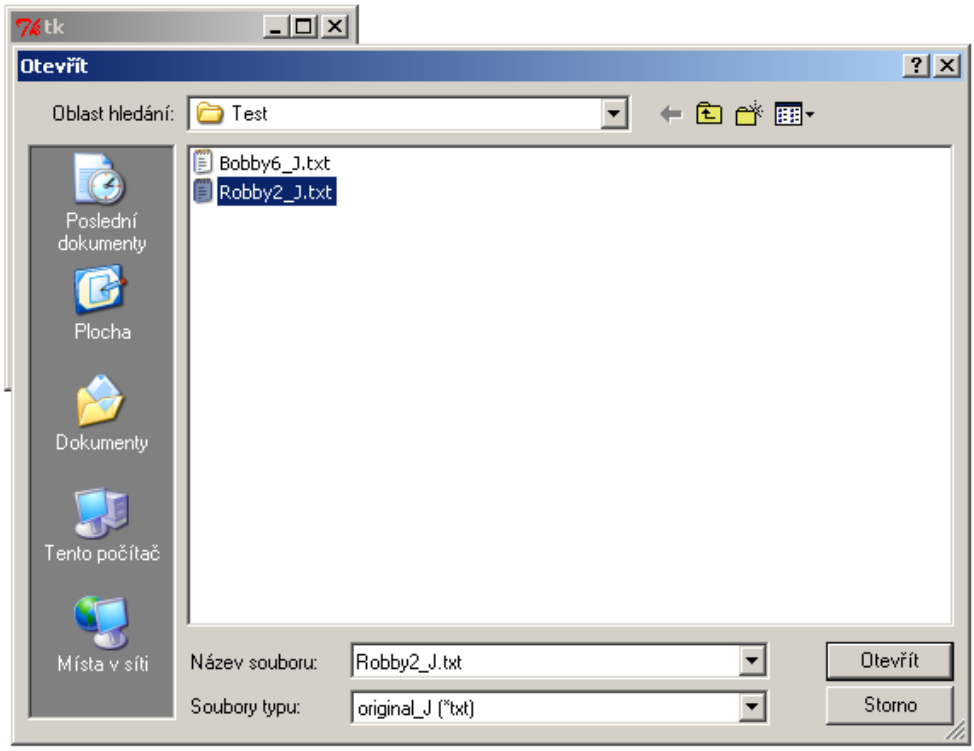
Step 2: Choose the original input file.



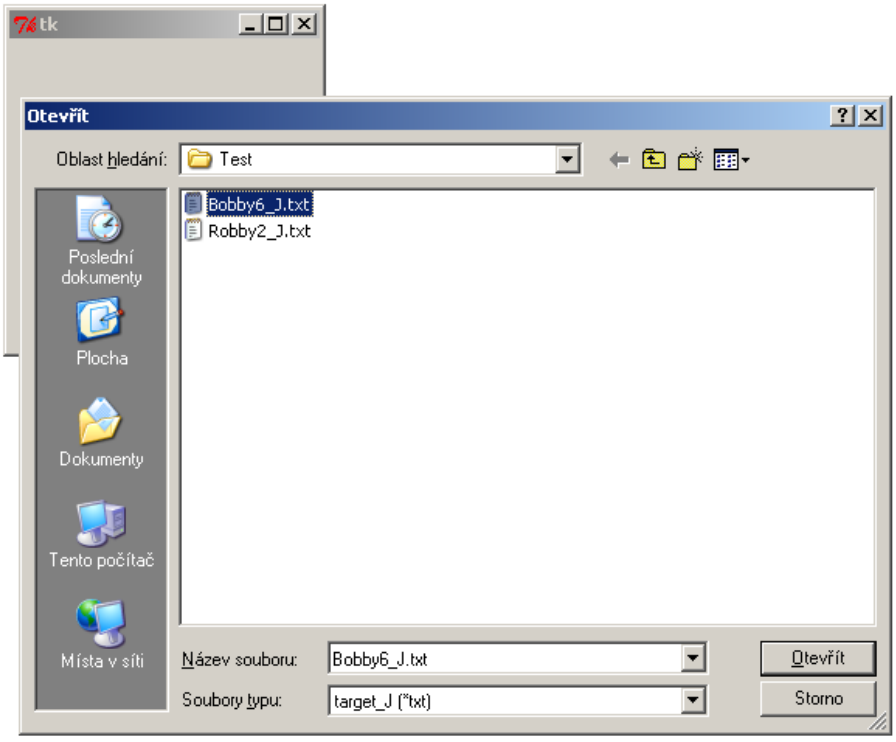
Step 3: Choose the original output file.



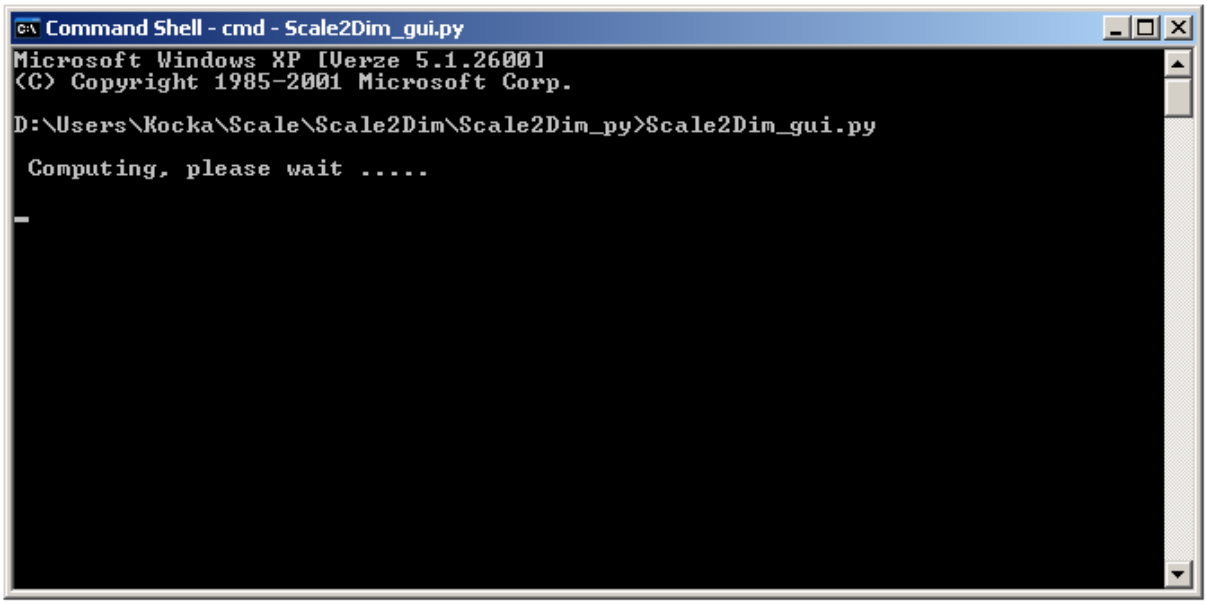
Step 4: Choose the target input file.



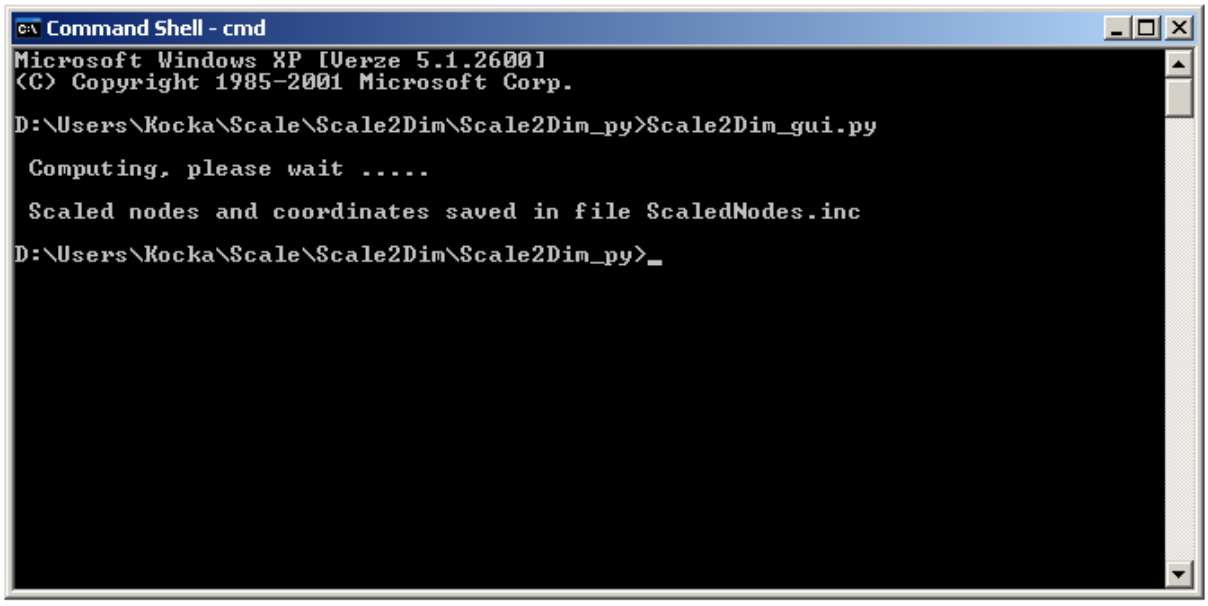
Step 5: Choose the original segment file.



Step 6: Choose the target segment file.



Step 7: Computing



Step 8: Finished

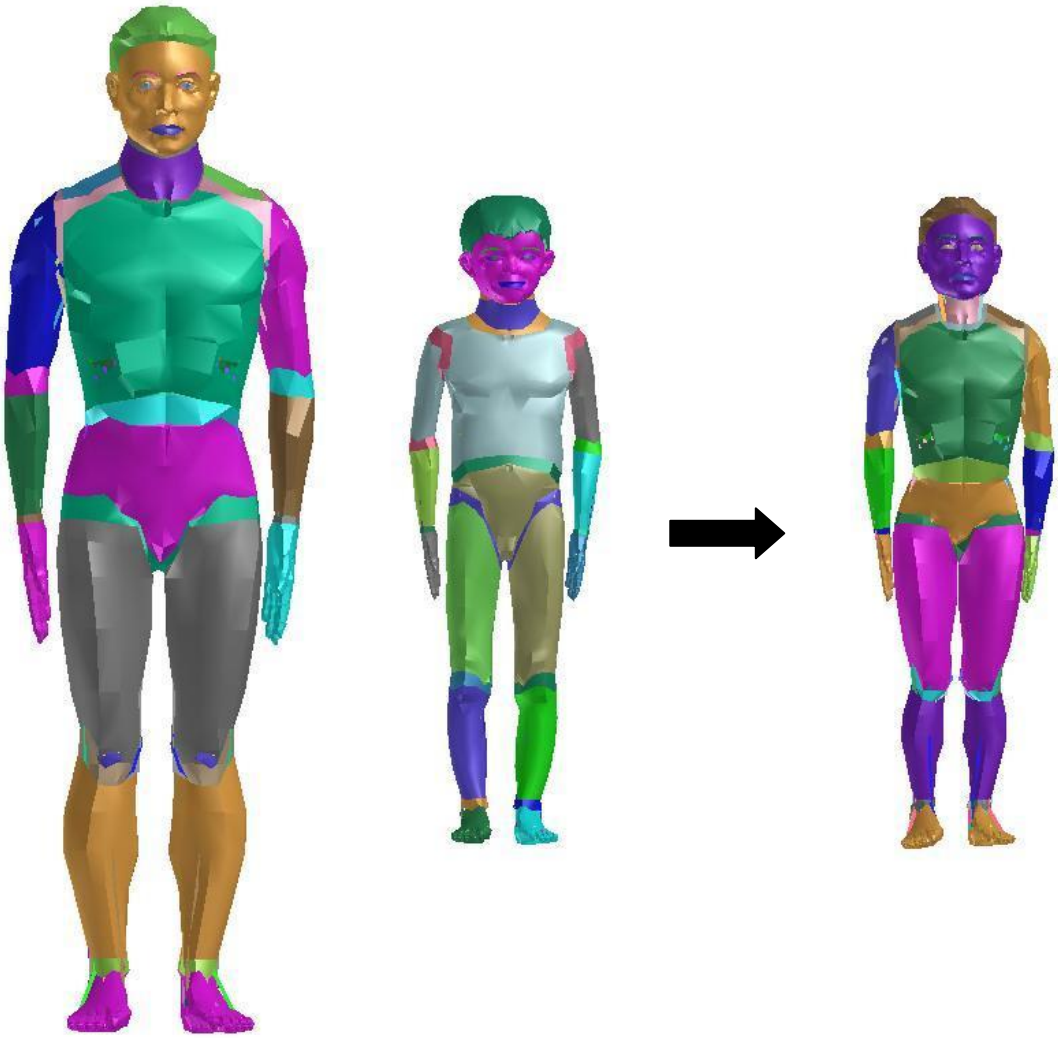


Figure 3: (Original) Robby's geometry transformed to (target) Bobby model.