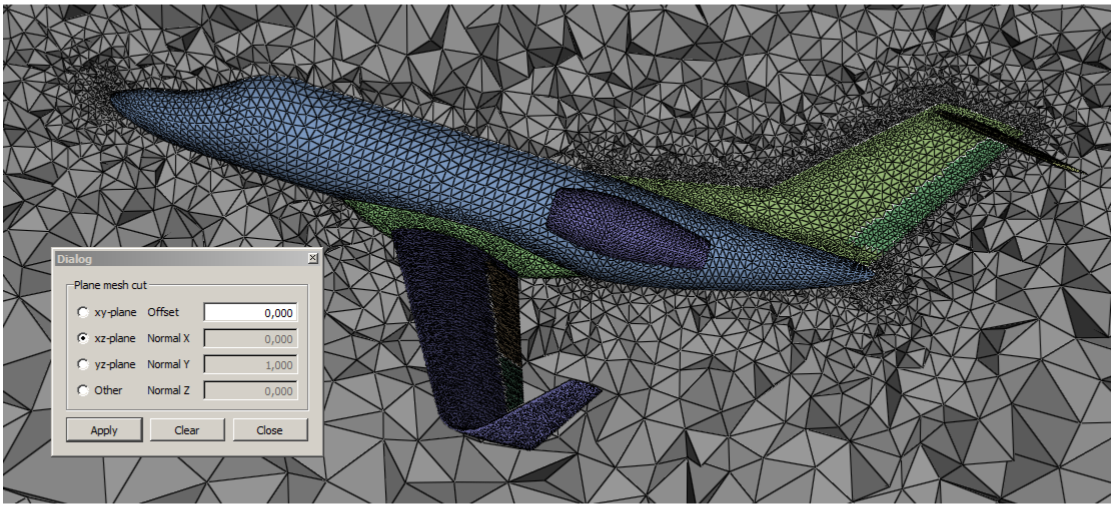
Aerodynamic Design of Aircraft with Computational Software - Supplementary Material

# User Guide - Sumo

David Eller

http://www.larosterna.com/




Aeronautical Department

Royal Institute of Technology

# Contents



# 1 Introduction

Sumo is an easy-to-use aircraft surface modeling system, highly specialized towards aircraft configurations. It is capable of rapidly creating aircraft geometries and automatically generating triangular surface and tetrahedral volume meshes. Originally developed following requests for a simple modeling system by the Swedish National Aeronautics Research Project DirSim (NFFP-4), it was sponsored by the EU project SimSAC for further development since 2008-2010. For more information, please visit http://www.larosterna.com/.

# 2 Installation

The latest `Sumo` version for your OS can be downloaded from the svn server or `http://www.larosterna.com.` The current `Sumo` version comes with an executable for `TeTGen` and one for `sumo's` mesh viewing program `dwfscope`. If you are using the VM provided as resource everything is ready to go. If not, visit `http://www.larosterna.com/` for instructions.

# 3 Main GUI

The GUI Window includes the menu bar and the icon toolbars at the top, the component tree at the left, View mode tabs and the main graphical window.

# 4 Import Geometry

Import CEASIOM a/c Builder xml file by "File - Import - Import CEASIOM file". `Sumo` can import its own native .smx files, the XML Ceasiom format from the SimSAC project, point grids from text files, or "overlay geometries" as .igs, .stp or .stl files. The `ceasiompy` environment installed in the VM has translators between .smx and the CPACS XML format. The CPACS format is more general than .smx so translation may for some models not be exact.

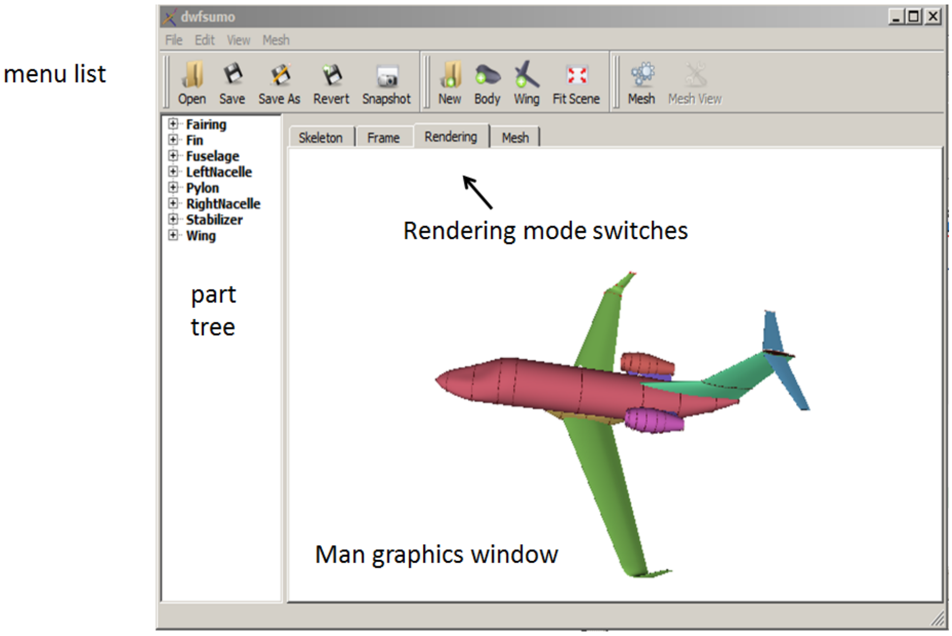



Figure 1: Main GUI Window

# 5 Geometry Modeling

## 5.1 loading the assembly

Load the geometry at start up with one of the built-in assembly templates or from existing .smx. This can also be done through "File - Open file", or the Open icon, see Fig.2.

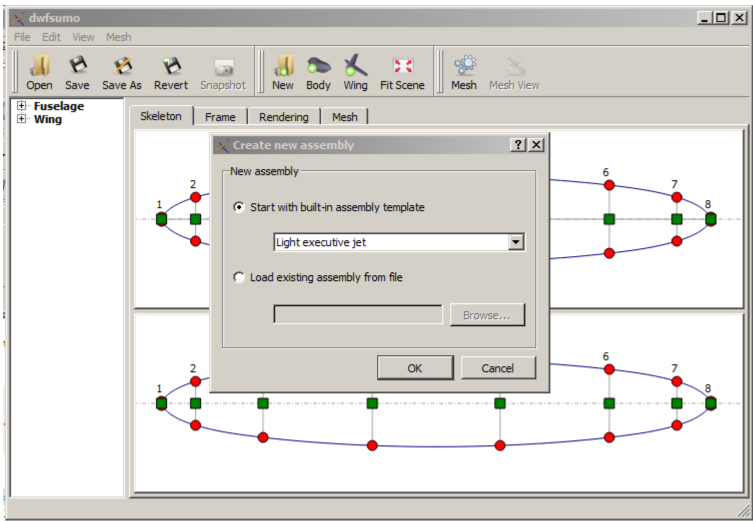
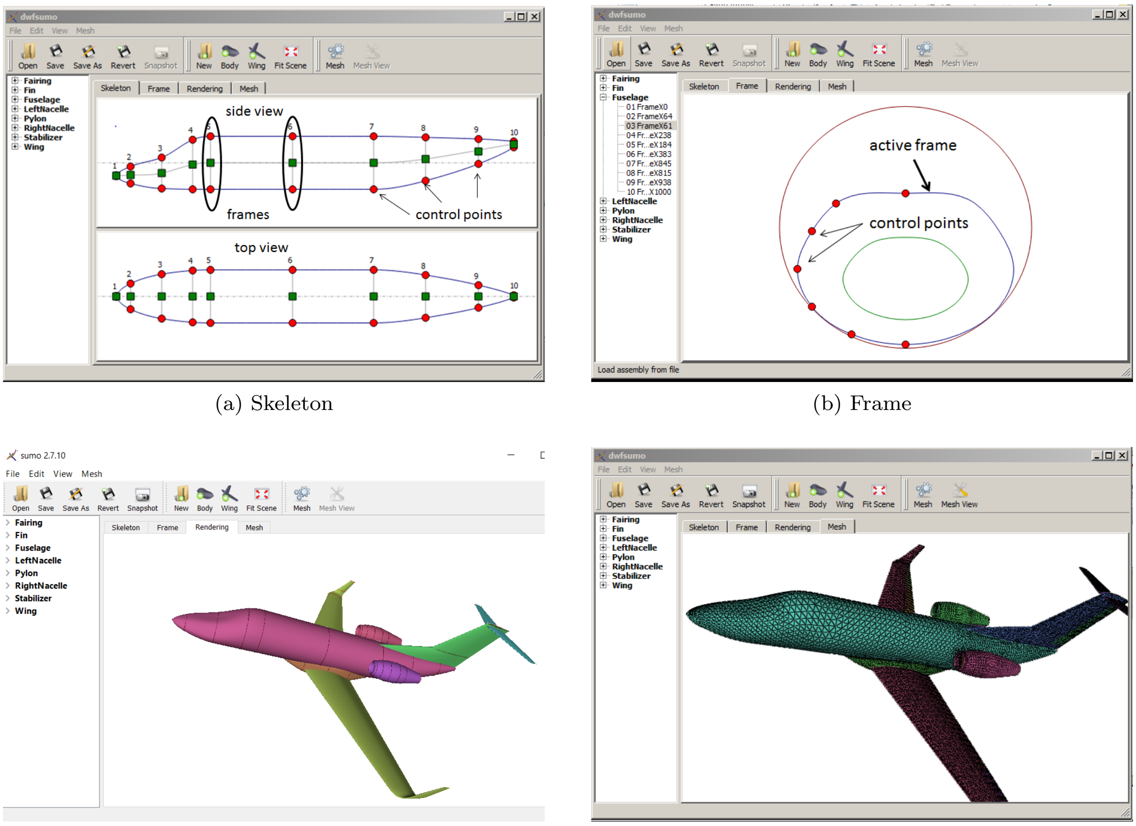



Figure 2: Loading Geometry

## 5.2 Viewing the Assembly

The View switch tabs - `Skeleton`, `Frame`, `Rendering` and `Mesh` showbody the geometry of the assembly from different perspectives, see Fig.3. "Skeleton" displays the distribution of the frames along their longitude direction, while "Frame" contains the detailed distribution of control points on each frame. Select another section from the part tree to the left to change to edit another frame. "Rendering" shows the assembled surfaces in 3D; press the left mouse key and move to rotate the view, and press the right mouse key and move to translate the view, and scroll the mouse wheel to zoom in or out.




(a) Skeleton (b) Frame

(c) Rendering (d) Mesh

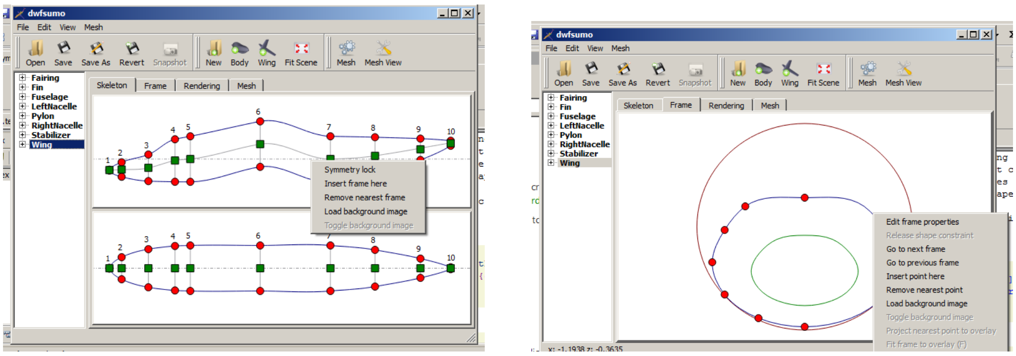
Figure 3: Viewing Geometry

## 5.3 Editing the Geometry

There are mainly two alternatives to effectively edit the geometry.

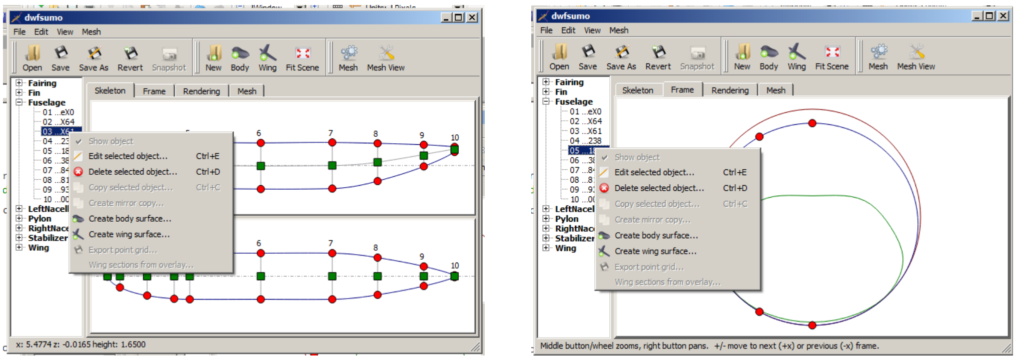
First, the frames and control points can be changed interactively in the main graphical window in the skeleton or frame mode. For example, in the *skeleton* mode, the current frame's center is moved by dragging the green square point ; new frames can be added by right clicking on the geometry and selecting "Insert frame here", while current frames can be removed by selecting "Remove nearest frame", see Fig.4a. In the *frame* mode, the red control points on each frame can dragged; also, red control points can be added or removed by selecting "Insert point here" or "Remove nearest point" on the right clicking menu, see Fig.4b. In addition, through the option "Edit frame properties" on the right clicking menu, the position of the frame's center and the shape of the frame can be modified. Special frame shapes such as circles, ellipses, and super-ellipses are available via "Edit frame properties - Special shapes" option.

Second, changes to assembly parts and frames can be made by right clicking in the part tree, through the "Edit selected object..." menu, see Fig.5.




(a) Skeleton (b) Frame

Figure 4: Editing Geometry through GUI




(a) Skeleton (b) Frame

Figure 5: Editing Geometry through Part Tree

Sumo uses the body coordinate system, as follows:

| Coordinate | Description |
|---|---|
| Origin | aircraft nose point |
| x | through nose and tail of body, pointing toward tail |
| y | perpendicular to x-coord, pointing toward the right wing tip |
| z | right-handed, perpendicular to x-y coordinates(pointing “upwards”) |

When editing the Wing component, the reference point for the tip section and central section are illustrated in Fig.7 respectively.

## 5.4 Editing the Control Surfaces

Control surfaces, jet engine and nacelle geometry are edited through the Edit menu.

## 5.5 Saving and Exporting the Geometry

The aircraft assembly can be saved via ”File - Save Assembly”, or be exported as IGES file by ”export to IGES”.

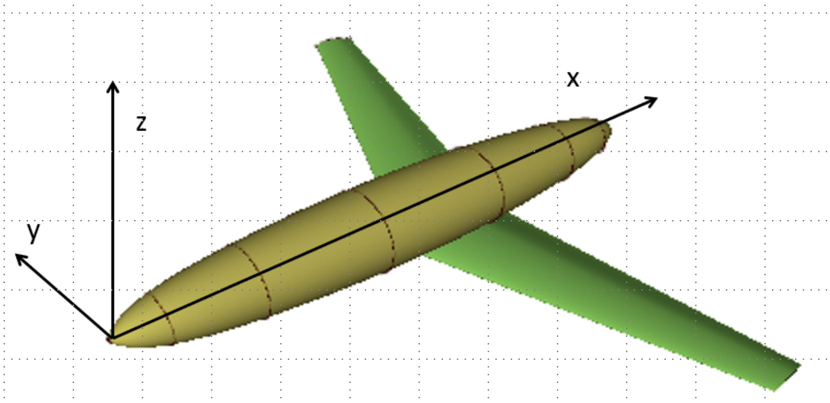



Figure 6: Body coordinate system used by Sumo

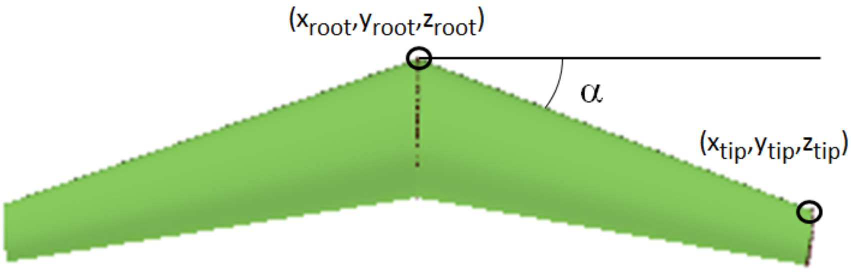


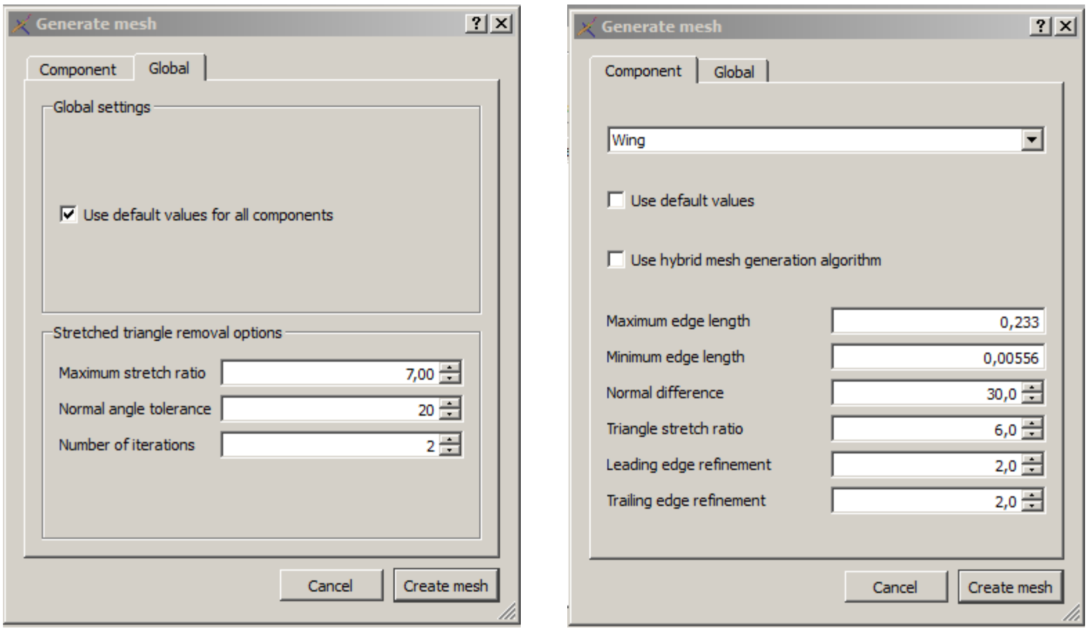

Figure 7: Reference point for wing tip and root sections

# 6 Mesh Generation

## 6.1 Creating the Surface Mesh

Sumo can generate high quality unstructured surface meshes with minimal user intervention. To generate the surface mesh, click "Mesh - Generate surface mesh". In generating the surface mesh, one can choose to either use the default values for the mesh control parameters or make manual changes. Fig.8a shows the global parameters setting panel, explained as follows:

- Select "Use default values for all components" to use the default values for all the components' mesh control parameters.
- *Maximum stretch ratio*: the triangle stretch ratio is defined as the ratio of longest to shortest edge. The magnitude of the maximum acceptable ratio depends entirely on the properties of the flow solver used. A high stretch ratio allows to discretize large surfaces with small regions with strong curvature, such as thin wings, with fewer elements. Triangular mesh elements with stretch ratios beyond the acceptable value shall be removed.
- *Normal angle tolerance*: the triangle normal angle is the angle formed by two neighbour normal edges (the line connecting two vertices of the 3D triangle). An over-stretched triangle usually accompanied by a small normal angle, is unfavorable.




(a) Generate mesh - Global parameters

(b) Generate mesh - Component parameters

Figure 8: Editing Surface Mesh Generation Parameters

Fig.8b shows the setting of mesh parameters for different components, the meaning of the parameters as follows:

- Select the component whose mesh parameters you want to modify.
- Select "Use default values" if you prefer the mesh control parameters' default values for the component, deselect otherwise.
- *Maximum/Minimum edge length*: the acceptable maximum / minimum edge length among the selected component's surface mesh elements. The minimum edge length serves to avoid resolution of irrelevant geometric details.
- *Normal difference*: the acceptable maximum angle between any two neighboring surfaces' normal vectors. The normal difference is small on a flat surface patch where normals are nearly the same everywhere.
- *Triangle stretch ratio*: the maximal ratio of the longest edge to the shortest edge accepted for the component's all triangle elements,
- *Leading/Trailing edge refinement*: The extent of gradual decrease of the maximum edge length from the value specified to a smaller value along the leading/trailing edge line.

## 6.2 Saving the Surface Mesh

When the surface mesher has finished, select "Save surface mesh" to save the surface mesh, or select "Volume mesh" to generate the volume mesh. After the surface mesh has been generated, the aircraft's cross section area distribution is also available through "Mesh - Area distribution".

## 6.3 Creating/Exporting the Volume Mesh

The unstructured tetrahedral volume mesh is generated by selecting "Volume mesh" once the surface mesh generation has been finished, or via "Mesh - Generate Tetrahedral volume mesh". Note that the TetGen executable must be located first. The volume mesh generation panel is as Fig.9, on which the control parameters are explained as follows:

- *Farfield radius*: the outermost boundary radius for the volume mesh. The volume mesh domain is delimited by the aircraft configuration surface mesh and this farfield boundary.
- *Tet radius/edge ratio*: maximum ratio of circumsphere radius to tetrahedron edge length
- *Min dihedral angle*: the acceptable minimum angle between the normal vectors of two triangles sharing an edge.
- *Max tet volume*: maximum element volume

First locate the tetgen executable ("First pass"). Then click on 'run".

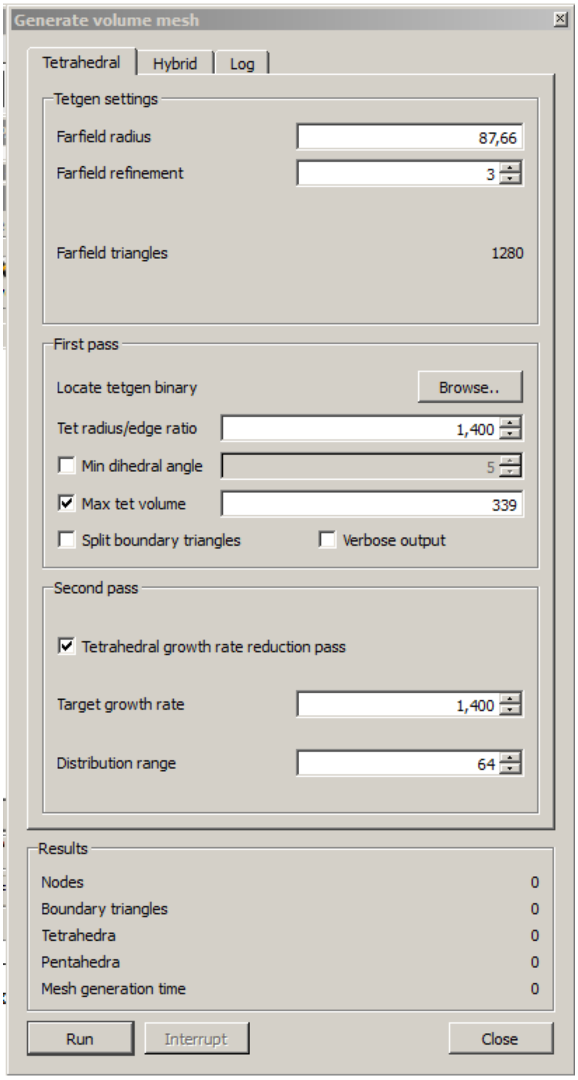



Figure 9: Volume Mesh Generation

The volume mesh has been generated successfully, when the line "tetgen terminated normally" is shown on the Tetgen log output. Close the panel and save the volume mesh by "Mesh - Save volume mesh". For viewing the mesh, save as .zml (native). Then start the `dwfscope` program, read the mesh file, and select a cut plane. The result on the light jet executive template with all default parameters appears on the title page. Select the desired output format, e.g. EDGE mesh file .bmsh or FLUENT mesh file .cgns.

# 7 Running Sumo in Batch Mode

Sumo may also be run in batch mode, which can be done through the cmd window or matlab by typing "sumo-batch somefile.smx". This will produce the following output files in the working folder.
somefile.igs [IGES file]
somefile.msh [surface mesh for dwfs]
somefile.smesh [input for tetgen]
somefile.bmsh [EDGE mesh]
somefile.aboc [EDGE boc file]

# A Sumo Input File Formats

Sumo input file .smx belongs to the XML structure family, which is used widely by many application programming interfaces (APIs) to transport and store data try “xml” on google. The basic building blocks for XML are elements, which can contain their own character content, attributes, and other elements, see Fig.10, which shows a section describing a control surface

```
<ControlSystem>
   <ControlSrf name="RightFlaperon" type="TEF" wing="MainWing">
     <Hingepoint chordpos="0.83" spanpos="0.037"/>
     <Hingepoint chordpos="0.83" spanpos="0.185"/>
     <Hingepoint chordpos="0.83" spanpos="0.345"/>
     <Hingepoint chordpos="0.83" spanpos="0.48"/>
   </ControlSrf>
```

Figure 10: .SMX file control surface element

As for Sumo, aircraft models are defined from the cartesian points on different sections, e.g. the point arrays forming fuselage sections along longitudinal axis, or the point arrays forming chordwise wing sections throughout the spanwise directions. Hence, the parameter value for each basic element consists of the point coordinate arrays constructing that element, see Fig.11.

```
<?xml version="1.0" encoding="UTF-8"?>
<Assembly ppMaxPhi="30" ppMaxStretch="5" ppNiter="1"
sumo_version="132865">
  <WingSkeleton flags="autosym,detectwinglet,cubic,"
name="MainWing" origin=" 2.4 0 0.15 " rotation=" 0 0 0 ">
    <WingSection airfoil="ah93k132" center=" 0.07 6 0.42 "
chord="0.82" dihedral="0.0698132" name="RightOutboard" napprox="-
1" reversed="false" twist="0.0406662" vbreak="false" yaw="0">
<![CDATA[ 1 0.00071
 0.97897 0.00192
 0.96313 0.00352
 0.94521 0.00573
 0.92646 0.00832
 0.90745 0.01114
 0.88859 0.01406
 0.87068 0.01711
 0.85388 0.02083
 0.83732 0.02571
 0.82 0.03169
 0.80183 0.03818
 0.78351 0.04477
 0.76535 0.05131
```

Figure 11: Sumo Input File

The components (element) in the input file may include Wing Skeleton, Body Skeleton, and Control Systems, each of which in turn is composed of many sections, with sectional position and properties defined in the attributes block.