Ecole Nationale des Ponts et Chaussées

2011-2012

Projet de Fin d'Etudes

Département Génie Civil et Construction

Jiří VRAJ

Elève double diplôme

Design of a general detail of cushion structures

Parametric study of clamping profiles and steel upstands for ETFE structures for all possible static actions

Projet realise au sein de LEICHT Structural engineering and specialist consulting GmbH

Königstrasse 9, D-83022 Rosenheim, Germany

20.02. – 22.06. 2012

Tuteur : Dr.-Ing. Lutz SCHÖNE

Acknowledgements

I would like to thank to all my family for kind support, as well as to whole company LEICHT Structural engineering and specialist consulting GmbH for giving me space and means to fully apply myself into this project.

Résumé

Conception des structures en membrane ou en coussin gonflable demande de profilés d'ancrage étant monté tout autour de ses frontières. En général, ces profilés suivent les axes d'une structure primaire. Mais comme ces structures ont tendance d'adopter des formes plutôt organiques, on envisage des problèmes de géométrie aux nœuds d'intersection de profilés. La géométrie complexe implique des problèmes de compatibilité, ainsi qu'un comportement structurel 3D.

Cible de ce PFE est de PARAMETRISER un tel nœud et, à la fin, donner à l'organisme d'accueil un outil pour pouvoir traiter les profilés d'ancrage vite dans les futurs projets.

Mots-clés : Paramétrisation, Panelization, Profilés d'ancrage, Interface, Torsion

Abstract

Design of membrane and inflatable cushion structures requires clamping profiles being mounted all along the cushion's edge. These profiles generally follow the primary steel structure. But as these structures often adopt a rather "organic" shape, we encounter various geometric problems in points where the clamping profiles intersect. As long as their axes intersect rarely in a single point, we encounter a delicate mix of 3D-structural behaviour of such element as well as geometric compatibility problems.

Aim of this internship should be to PARAMETRIZE such nodes and, eventually, give the hosting organization a tool to deal with it easily and rapidly in upcoming projects.

Keywords: Parametrization, Panelization, Clamping profiles, Interface, Torsion

Contents

List of Tables

List of Figures

List of Annexes

1 GENERAL INTRODUCTION

This project will concern a specific structural detail of light-weight free-form structures. This type of structures is in general case composed of a grid of slender elements, being filled by a lightweight material such as textile or ETFE membrane. Clamping profile for the filling occurs typically in hundreds of meters, accommodating all thinkable deformation to respect desired form. Objective of this study is to provide a tool, which other engineers in the office could use in design of clamping profiles and – based on input by architects – which would solve the most time-consuming tasks on an automatic basis.

Three main topics can be seen in the presented problem and after a common introduction this study will be divided respectively. First part will deal with geometrical topics with main issues being COMPATIBILITY and PARAMETRIZATION (p. 34). In the second part we will deal with INTERFACE between geometric model and FEM-calculation software to allow fast and robust interchange of data (p. 46). And finally, we should be able to run FEM analysis to optimise the whole structure, while the major keywords will be CONSTRUCTION and RESISTANCE (p. 55).

Knowing the time limits of this master thesis, main goal **is not** to elaborate all the topics up to their final shape and full operational state. It is rather to set a framework and bring the key-elements to life. Then this framework should be further fine-tuned and interconnected.

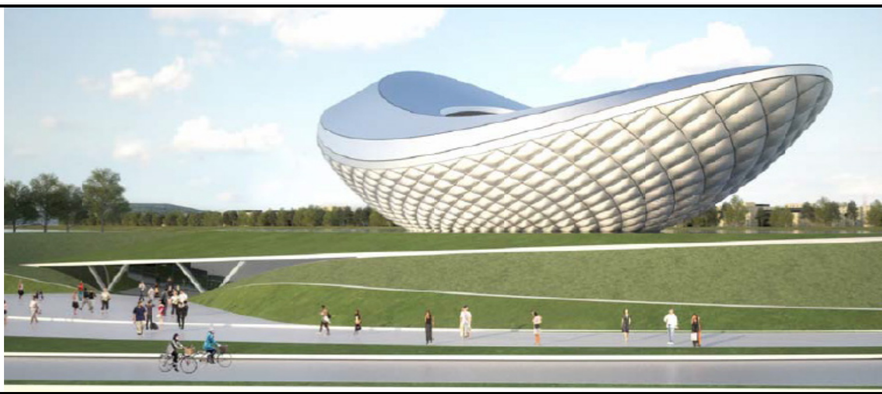


Figure 1-1: Gridshell	Grid structure with cushion fillings (Project of RIVER CULTURE PAVILION, South Korea)

2 PRESENTATION OF HOST COMPANY

2.1 The Company

My internship took place in the engineering office LEICHT Structural engineering and specialist consulting GmbH. It is a young company which found its field of activities mainly in the domain of lightweight structures. By these we understand textile membrane structures, inflatable cushion structures, lightweight facades etc. At the same time, LEICHT employs engineers specialized in "traditional engineering" of concrete and steel.

Originally it was a company based in Germany with two offices (Munich and Rosenheim), but in October 2011 a new subsidiary, LEICHT France, was founded in Nantes with lightweight and timber structures as its main interest.

2.2 Philosophy

Because of their expertise in many domains of civil engineering and structural design, the company assures a project management from alpha to omega. Many years of experience in national and international projects enable LEICHT to keep the overall process in mind at all times. So LEICHT ensures that each individual measure supports the overall success of the construction project. LEICHT measures the success of the projects and tasks it has undertaken in terms of esthetics, functionality, efficiency and sustainability - you can measure LEICHT by its success.

LEICHT supports and advises its partners in all matters involving structural engineering, especially in cases in which the building task or material does not allow for a standard solution. Besides support structure design spanning a variety of materials, LEICHT also takes over approvals in individual cases, general technical planning and monitoring. When it is requisite, LEICHT develops software, hardware and construction systems and carries out research on its own in its search for alternative solutions. Energy consultations, sustainability and life cycle analyses as well as appraisals and expert opinions complete the full-service performance range of the LEICHT experts.

2.3 Employees

LEICHT is a handpicked interdisciplinary team of experienced graduate civil engineers from the areas of practice, teaching and research, whose specializations cover the entire spectrum of structural engineering. Architects, technicians and experts in art history and monument conservation round off the engineering staff and enable the appropriate creation of custom-tailored project teams with a project manager as permanent contact and quality control at CEO level.

2.4 Board

Following his study of structural engineering at the University of Stuttgart Dr.-Ing. Schöne first worked at the chair for the History of Structural Engineering at the BTU Cottbus in the areas of teaching and research. As an engineer at Covertex he became an expert for ETFE films and lightweight construction in general. In 2007, he founded the LEICHT GmbH. He earned his doctorate with his dissertation on the History of Reinforced Concrete Shell Construction. As of July 2011, he is publicly appointed and sworn expert for structural steel engineering and membrane construction.

Marcel Enzweiler studied structural engineering at the University of Applied Sciences Munich where he was subsequently entrusted with duties as a lecturer at the chair for Structural Design and later at the chair for Support Structure Design for Architects. Starting from 1993, as structural engineer at the engineering office of Behringer and Mueller, he was responsible for numerous projects in the area of surface construction before he joined the management of LEICHT in 2009. As early as 2008, he was included in the list of particularly expert engineers for timber, steel and solid construction.

Florian Weininger studied structural engineering at the Technical University of Munich. In an in-depth study of statics and timber construction he became a specialist in the field of construction engineering before he expanded his expertise at Covertex with regard to membrane and film construction in order then in 2007 to found the engineering office LEICHT together with Lutz Schöne. Besides his activities for LEICHT, he teaches and conducts research at the chair for the Science of Support Structures of the faculty of architecture at the University of Applied Sciences Munich.

3 DEFINITIONS

3.1 Description of Freeform Structures

In this chapter we will deal with description of free forms starting with their geometric properties and focusing to our cases of interest in the end.

3.1.1 Classification

Freeform structures can be classified with respect of various features and properties. In the following sections different criteria and features of geometry of free forms will be discussed. See Annex I.

3.1.1.1 Curvature

Single-curved surfaces have a line in common with a tangent plane, whereas double-curved surfaces have only one point in common with a tangent plane.

3.1.1.2 Developing into a Plane

By definition, the only surfaces which can be developed into a plane without any distortion are those single-curved. To bring a double-curved surface on a plane, they have to be either projected or discretized and then developed.

3.1.1.3 Generation of Surface

Free forms can be generated in different ways. The major ones are listed below. See Annex I.

- **Surfaces of Revolution**
  - Generated by a curve rotating about an axis.
- **Translation**
  - Generated by a curve being translated along one curve, or along a different curve at each end.
- **NURBS Surfaces**
  - Surfaces generated from control points and their weight. They are based on theory of NURBS (Non-Uniform Rational B-Splines), whose main benefit is fact that they can be exactly expressed in mathematical sense and solved with mathematically stable way. NURBS represent a tool by which an arbitrary form can be exactly described, recorded and reproduced (Sederberg, 2010).

3.1.1.4 Structural Action

Behaviour of a free form depends on its shape. If the form follows an optimum shape (minimal surface, catenary) then tension-compression forces are predominant. On the other hand, if the form is rather one rather synthetic (translation, revolution, NURBS), then neither the internal forces are optimal. Following diagram will help us to understand how free forms work from a structural point of view.

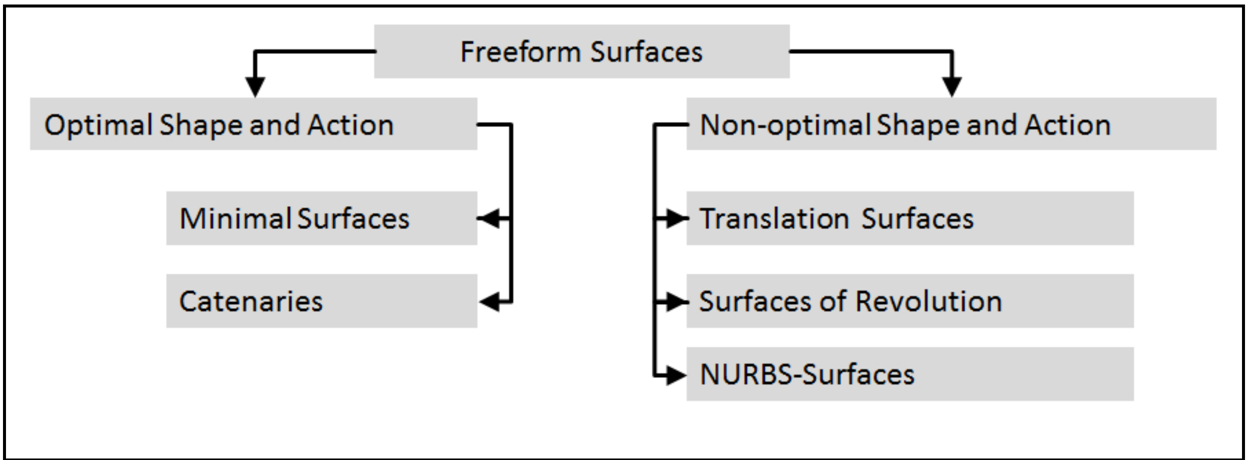


Table 3-1: Freeform Classification	Single-curved surface (left) and its developed shape (right)

3.1.2 Membrane Structures

Its light weight and ease of shaping is a main reason why these structures tend to be applied in freeform architecture. By membrane structures we mean such structures where the envelope is realised by means of a **single layer** of thin material of negligible bending stiffness. Their shape is given by supports and prestress. Their nature thus implies that they resist to loads by change of form and prestress and as such they are ***Form Active***.

3.1.2.1 Material

In general, for membrane structures is used some kind of textile membrane with woven core and protective sheathing, which provides protection against UV and water tightness if demanded. The way it was woven determines stronger and weaker direction of such fabric – WARP direction being stronger and FILL (or WEFT) direction being weaker. Due to orthotropic characteristics, prestress can be different in the two directions.

Strength and relaxation properties are tested by biaxial tests. It is important to say, that relaxation properties play an essential role in design and cutting-pattern generation as it tends to change stress-distribution in long term.

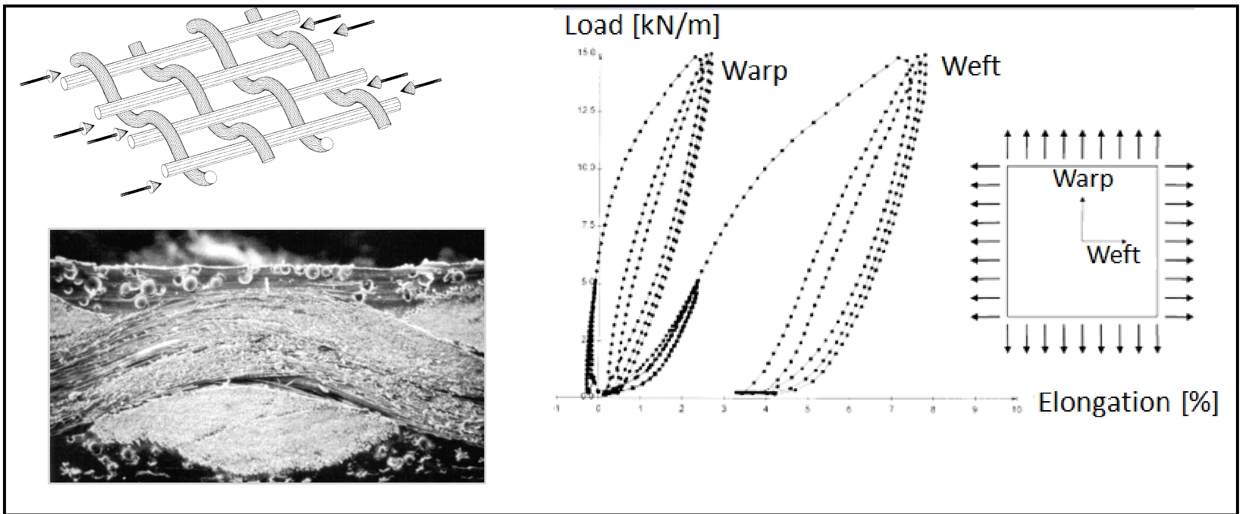


Figure 3-1: Warp&Weft Warp direction – white fibres; Fill (or Weft) direction – grey fibres (upper left)
Warp fibres in cross-section, Weft fibres in longitudinal section (lower left)
Biaxial test results (right) (LEICHT, 2008)

Weaving material has to provide structural strength and stability in time. In the following table one can find available weaving materials with their usual protective sheathings. In Annex III. you can find their mechanical properties and further details into this topic.

	Fabric	Protection
Unprotected Fabrics	Cotton	-
	Linen	-
	PTFE	-
Protected Fabrics	Polyamide	PU
	Polyester	PVC
	Glass	PTFE/Silicone
	PTFE	FEP
	Aramid	PCV
		Elastomer

Table 3-2: Materials Weaving and protective materials

3.1.2.2 Forms

Membrane can be fixed either in singular points, on the edge (either directly or by means of cables) or can be supported by means of a linear support. This determines its possible shapes (pictures in the Annex II.):

- **Saddle forms**
- **Ridge and valley cables**
- **High points, deep points**
- **Linear supports**

3.1.3 Pneumatic Structures

Pneumatic structures resist external loads by prestress as well as membrane structures, but this prestress generated by overpressure of gas on the inside. If they are single-layered, then the whole interior space of such building has to be pressurized.

Figure 3-2: Pneus Single-layer pneumatic structure from the outside (left) and the inside (right)

But pneumatic structures can also be multi-layer, where the overpressure is closed by two or more layers of a membrane, in a form of a cushion.

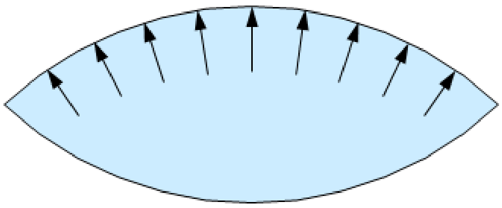


Figure 3-3: Cushion Double-layer cushion

From a technical point of view, pneumatic structures can be seen as a sub-set of membrane structures and as such they are subject to processes of formfinding, division into strips, flattening of the strips and compensation.

In addition to membrane structures (in sense of chapter 3.1.2), we have to think of an air-supply system as well as an emergency system in case of deflation. In such case, waterponding may occur and its solution represents an interesting engineering problem. In the name of clarity, this detail will not be developed further in this thesis.

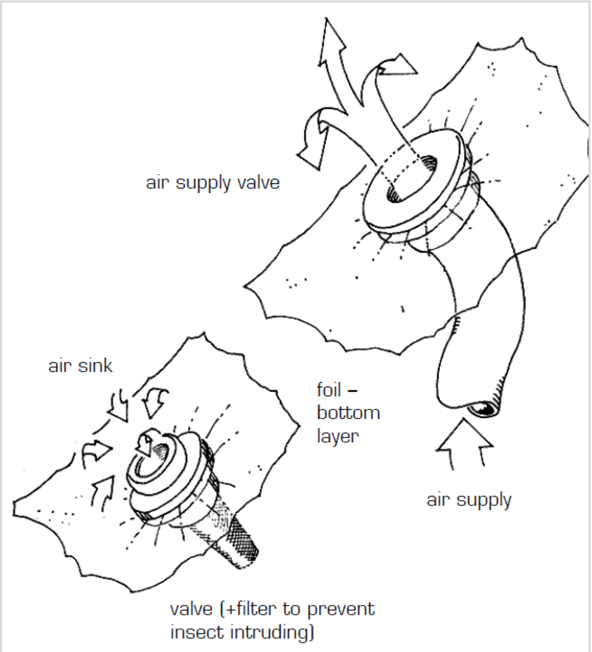



Figure 3-4: Cushion details	Air supply scheme (left); application to real structure (right) (LEICHT, 2011)

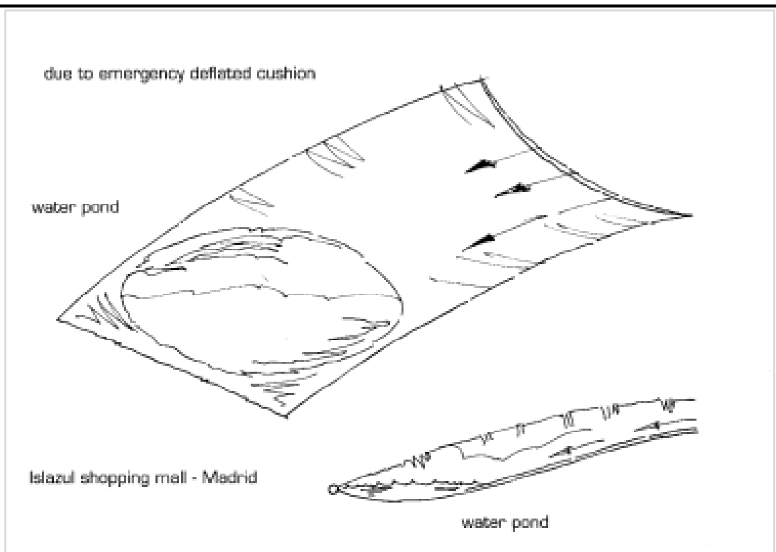


Figure 3-5: Waterponding	Accidental deflation of a double-layer cushion creates a water pond and considerable overload of the structure. (LEICHT, 2011)

3.1.3.1 Material

As for materials, we use different kinds of ETFE foils with various degree of translucency, colour and pattern. Typical thickness is in order of 200 µm and density of 1,75 g/cm^3. See Annex III for some further details.

3.1.4 Formfinding and Calculation

As for a form active structure, prior to construction we have to find its equilibrium form under prestress and self-weight in a process called ***Formfinding***. There are currently two methods for Formfinding and these will be briefly described below.

3.1.4.1 Analytical Formfinding Based on Force Density Method (FDM)

This method discretizes a membrane into "cables" and then, using Force Density Method, the equilibrium shape is analytically found in one iteration step. Force Density stands for ratio of cable-force divided by cable-length. This method best suits non-sheathed membranes with close-to-zero or zero shear strength.

3.1.4.2 Formfinding Based on Minimal Surface (soap-bubble model)

Given prestress and support conditions, this model tends to find such equilibrium, where stress would be equal in both directions. Typically this behaviour is presented on a model of soap bubble. In pioneering times of membrane structures, these studies were priceless. This approach best suits sheathed fabrics with non-zero shear strength. Because stress-homogenization in long term, isotropic materials converge to Minimal-Surface equilibrium rather than to the one calculated by FDM (Moritz, 2007).

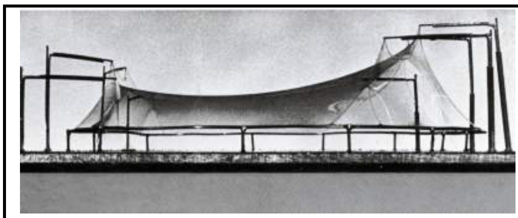


Figure 3-6: Soap bubble	Soap-bubble model (left); A picture of a real structure (right) (Otto, 2009)

3.1.5 Cutting Patterns

In order to be able to manufacture a membrane structure, cutting patterns have to be generated. Result of formfinding is divided into strips corresponding to single pieces of membrane material, creating seam-lines and these strips are then flattened. Due to material-economy, one of seam-lines follows a geodesic line, so that after flattening it is straight and can be aligned with edge of material (provided in form of rolls).

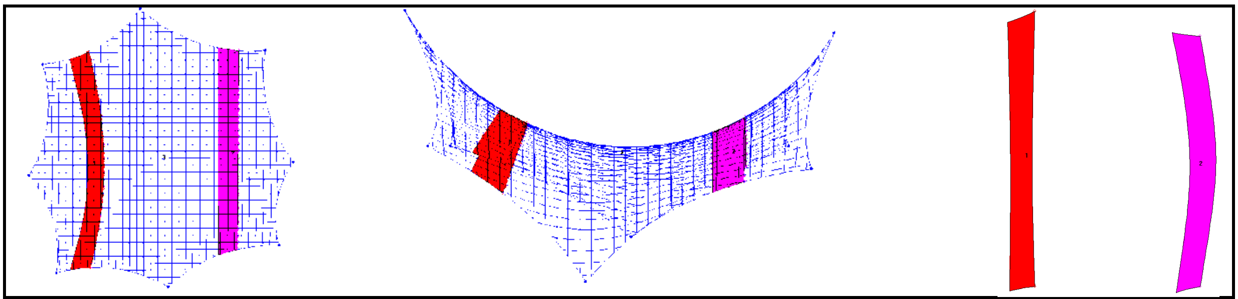


Figure 3-7: 3D/2D strips — Red strip follows geodesic line, pink one is straight in 3D
Top view of 3D-strips (left); Front view of 3D-strips (middle); Flattened strips (right) with red strip having one straight edge and pink being curved. (Weininger, 2010)

Based on biaxial tests of chosen material, **compensation** is set. After the flattening procedure, we transform the geodesic line, which is now a straight line, parallel to the y-axis and the direction of warp (=y-axis) and weft (=x-axis) is given. With special factors of compensation in warp and weft we are able to change the border line. This ensures, that even after relaxation, the prestress in membrane will not disappear (Weininger, 2010).

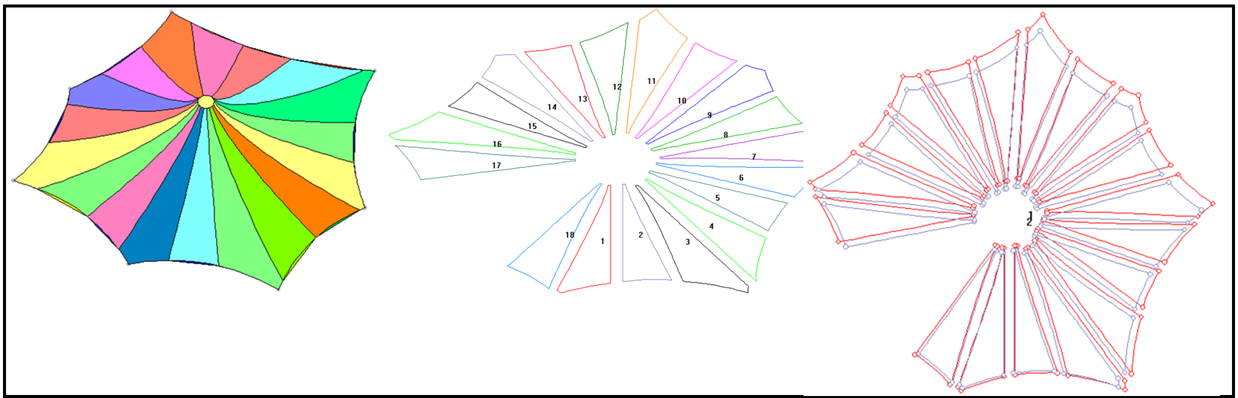


Figure 3-8: Cutting patterns — 3D Formfinding result (left); Flattened strips (middle); Flattened strips (red) and Compensated strips (blue) (right)

3.1.6 Constitutive Elements

In following section we will focus on structures using membrane as cladding, because such family of structures is an object of this thesis.

Starting from the inner side towards outside, first we generally find some **primary** steel or timber structure, which transfers all the forces toward foundations. **Upstands** are then attached to this skeleton, following a regular spacing pattern. These upstands form a support for **clamping profiles**, often referred to as **extrusions**, as they are extruded of aluminium alloys. Clamping profiles meet each other in **nodes**. Then the **Filling** can be eventually stretched in between extrusions. This can be either single-layer or multi-layer (inflated cushions) and is clamped in clamping profiles by means of **keders**. Keders are present at all borders of each filling panel and in order to allow installation and water-tightness, their **system-lines** must form a closed polygon.

When the membrane installation is finished, the **capping profiles** are then screwed on top of clamping profiles assuring watertight top surface. In the following points all highlighted elements will be described and developed.

- **Geodesic Line**
  - Curve belonging to a surface and connecting two points on the same surface, where its length is minimal. In other words, the shortest path between two points on a surface (see Figure 3-7 for its importance and interpretation).
- **System Line (SL)**
  - This is a line representing any geometry to which it is assigned. If not stated otherwise, it is located in its centroid. System lines will be referred to as "SLs".
- **Primary Structure**
  - Gives a main shape and forms a support to all ulterior structural members. Although overall appearance of primary structure might be that of a curved surface, individual links can be straight.
  - As the membrane-reactions are coming through upstands, primary structure is loaded by point-loads.

- **Upstands**
  - Upstands are short elements that allow a connection between a primary structure and a clamping profile. If ever the primary structure is rectilinear and clamping profiles are supposed to be curved, it is the individual upstands that must gap a variable distance between the two elements in question.
  - With respect to their position, they can be subject to various static actions. If they are located on a border of whole structure, a severe bending towards filling can be expected. On the other hand, upstands located inside are, in general, less loaded.
- **Clamping Profiles (Extrusions)**
  - It is this element that the membrane is attached to. Generally it is an aluminium extrusion of a special cross-section, so that it can accommodate membrane on its sides and a capping profile on top.
  - In analogy with upstands, those extrusions that are located on a border of whole structure are loaded more than those in the middle. As they are directly connected to membrane, extrusions are loaded by linear loads.
- **Filling**
  - This can be either a single-layer membrane or (perforated) aluminium sheet or as well a multi-layer inflatable cushions. No matter which material is used, in either case every single filling field has a keder on its circumference and this must continuous.
  - Obviously, filling is loaded directly by climatic loads and prestress. Prestress can be either mechanical (single-layers, or intermediate stressed layer in cushions) or pneumatic (overpressure in cushions).
- **Keder**
  - A keder is a circular reinforcing rope inserted in an edge-sleeve of every cushion or single-layer membrane field. This sleeve with keder sewn in is than clamped into a clamping profile and thus it cannot pop out. *Keder SLs* are taken as *Filling SLs*.
  - Keder, in effect, transfers reaction forces from the filling to the extrusions. Tests are executed to evaluate maximum resistance of filling material with respect to tearing the keder out.

- **Capping Profiles**
    - When filling is fixed in the clamping profile, this clamping profile is then covered by a capping profile from the exterior side to assure water-tightness and better thermal insulation.
    - It is not considered as a load bearing element.
- **Node**
    - A Node in general is a region, where constitutive members of a structure intersect. When mentioned in context of primary structure, it can be viewed as a single point, i.e. intersection point of member-axes. On the other hand, when mentioned in context of clamping profiles, it is rather a set of points of intersection, where keder-SLs meet.
- **Surface Panelization**
    - With respect to architects wishes the fundamental freeform shape can be panelized into more or less regular polygons of 3 and more vertices with straight or curved faces. See Figure 8-3 in Annex I.)

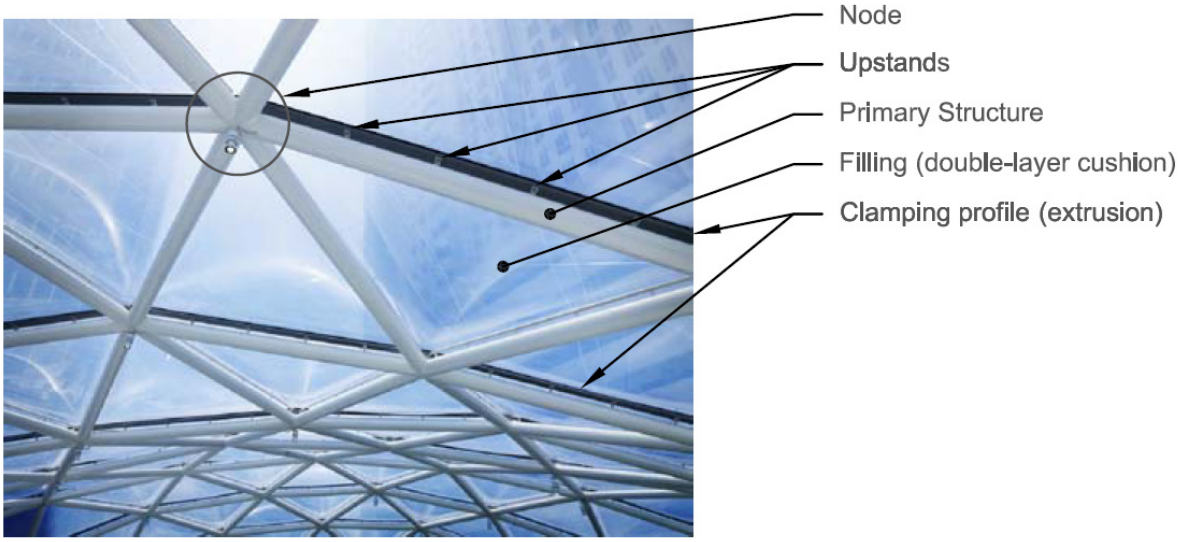


Figure 3-9: Free form detailed	Constitutive elements of a freeform structure. Mont Kiara Retail Mall, Kuala Lumpur, Malaysia

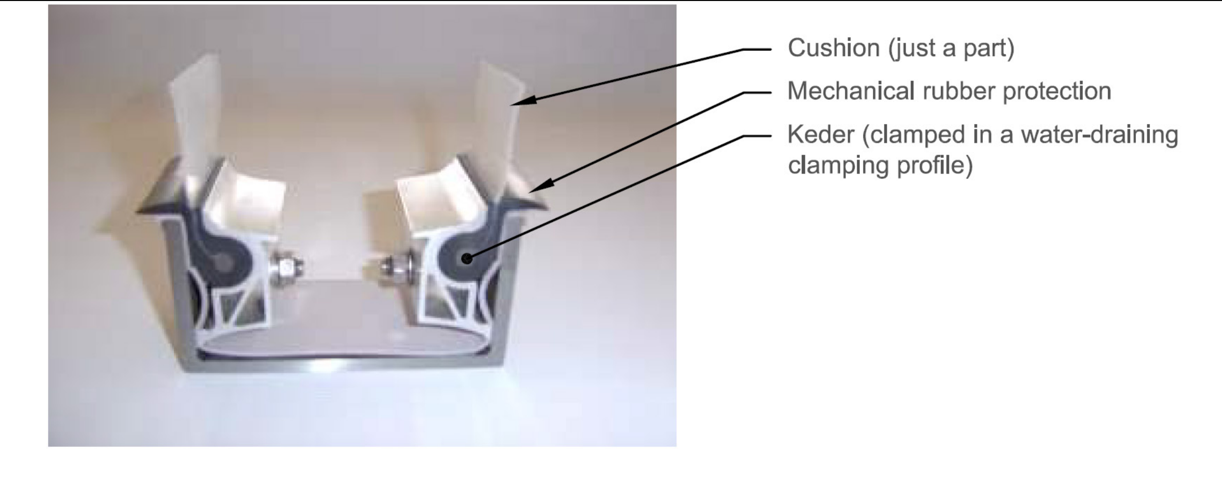


Figure 3-10: Gutter extrusion	Constitutive elements of a freeform structure (LEICHT, 2007)

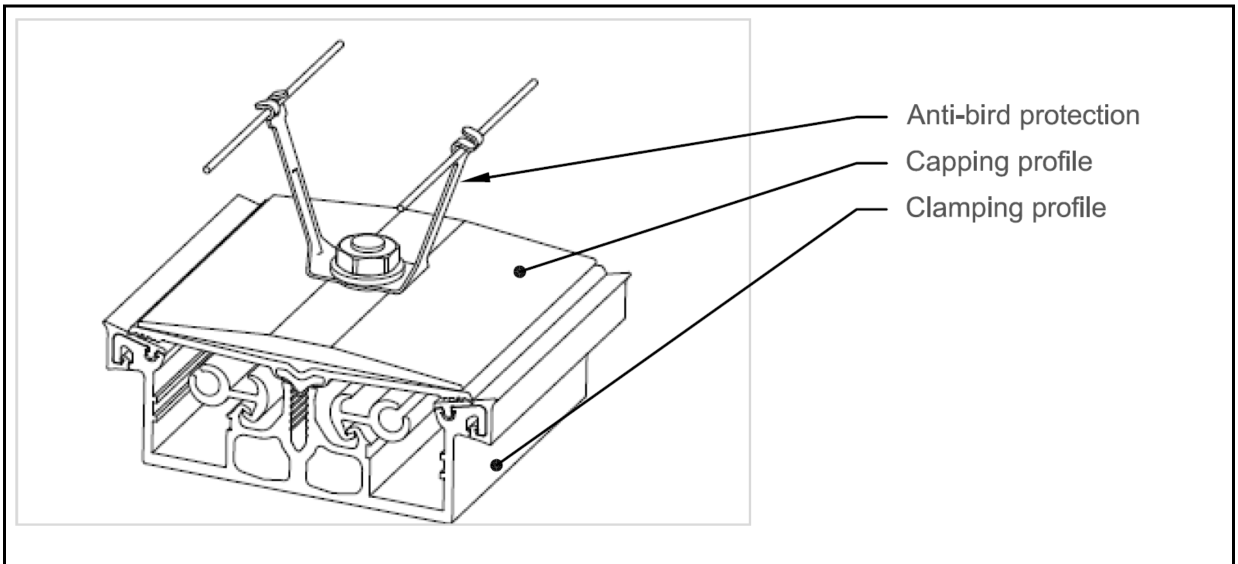


Figure 3-11: Extrusion - section	Section through typical clamping profile (LEICHT, 2007)

3.2 Definition of GEOMETRIC Problem and its Objectives

Overall objective of this part of work is to come up with a geometrical concept, which would be as simple as possible both for planning as well as for workers on the site. Only such solution can provide time-effectiveness of whole project with consistent quality of final construction.

Obviously, proposed solution to this problem must be generally applicable to any structure, where clamping profiles are involved. This request implies a "parametric" approach which would minimise repetitive extra-work necessary for each project. Last but not least, final solution should not be limited by type of panelization or other geometric features of a structure. It should work with high accuracy for any basic polygon, as well as for convex and concave portions.

Evolution of this problem represents most part of time spent with this thesis. At LEICHT nobody has ever dealt with this problem before in a complex way, so it was a start from a scratch.

We are basically looking for a **concept** – a "road" that will lead to final and most universal solution. While looking for it, we have to keep in mind following list (Table 3-3) of choices that have to be done. It is apparent, that answer to one question **MAY** influence another, thus it is impossible to find a correct starting point.

3.2.1 Problem of Nodes

The nodes have to follow geometric restrictions coming from above-mentioned definition – keder SLs must be continuous in the corners to form closed polygons. Objective of this point is to find such concept of node that would allow it. Subsequent problem is to find a solution for how to cap such node so that it was watertight. See Figure 3-12 for illustration.

STRUCTURAL GEOMETRY

Degree of curvature to treat
- Single-curved?
- Double-curved?
- Any?

Basic polygon to treat
- Triangles?
- Quadrangles?
- Higher?
- Any?

UPSTANDS

Length
- Equal for whole structure?
- Unique for each extrusion?
- Unique everywhere?

Type
- Adjustable length?
- Adjustable head?
- Several fixed types?

EXTRUSIONS

Torsion
- Straight?
- Twisted?
  - Pre-shaped?
  - Forced torsion?

Relation to prim. str.
- Parallel?
- Skew?

Axes
- Intersecting at nodes?
- Or only cushion SLs intersect?

NODES

Geometry
- Flat?
- Irregular?

Capping
- Flat polygon?
- Rubber plate with rigid cover (compensating non-planarity)?
- Unique shaped piece?

Table 3-3: Choices to make	Sum-up of choices and questions to be held in mind in order to set up a **concept**

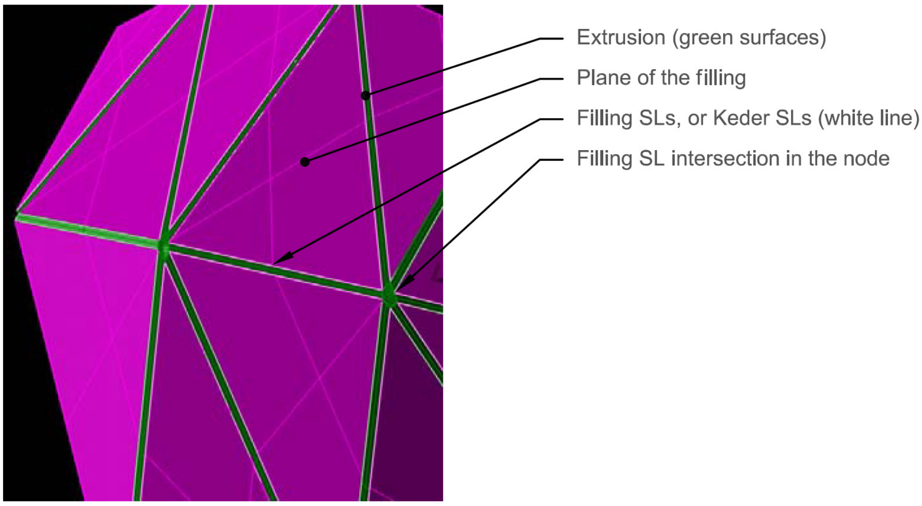


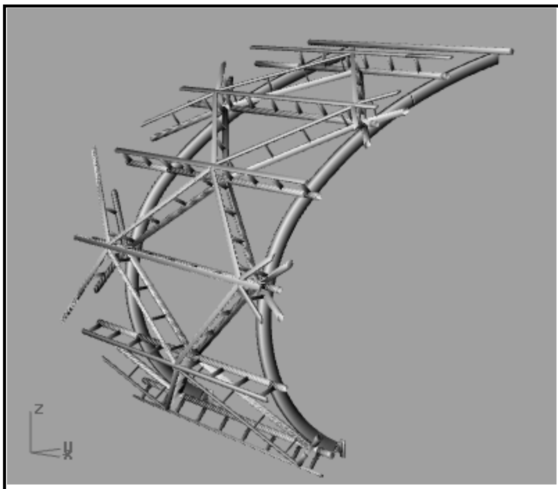
Figure 3-12: Problem of Nodes	Key elements of "Geometric Problem"

3.2.2 Problem of Clamping Profiles (Extrusions)

Extrusions are supposed to follow the primary structure while connecting individual nodes. Depending on structure panelization, the extrusions can be subject to bending in both axes and to torsion (as in Figure 1-1). If, for instance, there is a request of equal-length upstands, then extrusion has to follow a cylindrical surface. In practice, this means that an extrusion can be defined by two axes with constant distance in between (see Figure 3-12 – white lines).

3.2.3 Problem of Upstands

Understanding the relative position of extrusions and primary structures, it becomes also apparent that geometry of upstands has its specifics as well. An upstand has to bridge a gap between primary structure and extrusions. If these two elements are not collinear or if the primary structure is rectilinear and extrusions are curved, then upstands can be slightly rotated about primary structure or they can have variable length or their upper surface has variable angle with respect to its longitudinal axis.



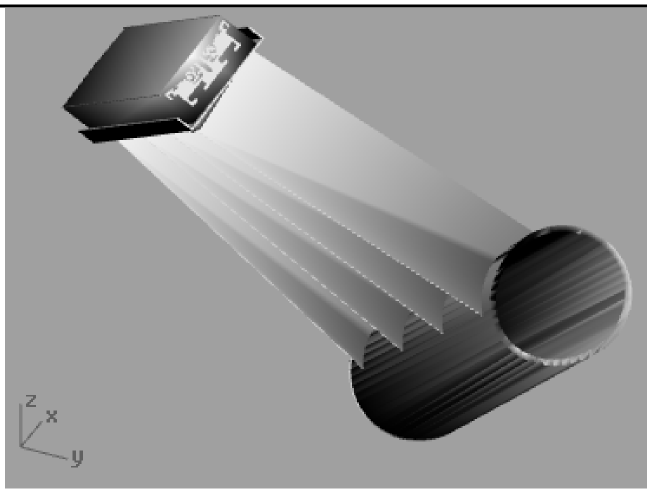


Figure 3-13: Problem of Upstands	Problem of Upstands: Overall view of a segment of a façade (left); Detail of upstands rotated about primary structure to provide support to an extrusion (right). Project by LEICHT at Franhofferstrasse, Munich

3.3 Definition of INTERFACE Problem and its Objectives

Now that we have all geometric data along with ways to analyse them, it is time to establish an interface between them. Such interface is expected to fulfil following demands:

- Easy (and direct) import of all geometric data from geometric model into FEM software. This must include coordinates of extrusion SLs and torsion data belonging to each of them. Format of choice is a text file.
- It should include a deciding mechanism, which will mark "low" torsion rates as a loading case, whereas "high" torsion rates as input geometry, based on previous experience. This preliminary sorting would be good enough for first iteration in FEM software.
- After thorough analysis of the extrusions in FEM software, it should be able to tag those that must be shaped in shop and those that can be shaped on site, by assembly.
- Upstands should be tagged at this step as well as they may be of completely unique shapes.

3.4 Definition of STATIC Problem and its Objectives

Having all the geometric data ready, it is now time to process it. Main points to observe will be, again, broken down below.

3.4.1 Problem of Nodes

Static analysis in nodes would focus mainly on thermal expansion of extrusion, while observing sufficient spaces between extrusions and interaction with node-capping.

3.4.2 Problem of Filling

Having extracted the keder-SLs and having set the section properties of extrusion, we will be able to analyse the filling as well. As long as membrane-structures are form-active and dominant stress is bi-directional tension, they generate reaction forces along whole length of its border. And these are essential for analysis of clamping profiles.

Ultimate goal of filling-study is to generate cutting patterns and seam-lines. Filling-panels can be then fabricated in warehouse using these data and shipped to site for assembly.

3.4.3 Problem of Clamping Profiles (Extrusions)

As stated in Chapter "Definition of GEOMETRIC Problem and its Objectives", extrusions are subject to the two bendings and torsion. Should the deformations be **"too high"**, the extrusions have to be shaped in a shop, before going to construction site. But if the deformations are **"low"**, they can be shipped to site as straight pieces. These limits between "too high" and "low" have to be established by a rigorous static analysis.

Extrusions have equal section properties throughout whole structure and, clearly, not all of them are equally loaded. So, after applying all possible reaction forces from filling-panels, reserve in resistance can be used to accommodate implied deformation. If the reserve is too low, the particular extrusion must be shaped prior to shipping to site.

That is to say, that torsion data has to be considered as **a load case** for some extrusions and as an **input geometry** for the others. Easy switching between the two senses must be possible.

3.4.4 Problem of Upstands

Tricky point of upstand calculation is that they can be all of unique height and different loading. This implies that rather than judging one critical unit it is going to be a check of a set of upstands.

4 GEOMETRIC PROBLEM

This chapter will deal with solution to all geometric problems and its ultimate objective is to provide all necessary system-lines with all additional geometric data for further analysis.

4.1 Methods

It was decided, that the most convenient option for parameterization is "Rhinoceros" with scripting plug-in called "Grasshopper".

4.1.1 Rhinoceros 3D

In full name "Rhinoceros NURBS modelling in Windows" this software is written by Robert McNeel & Associates (Robert McNeel and associates, 2010). Following advantages caused recent popularity of this software and were also a reason for choosing them for this thesis:

- Very competent NURBS-engine allowing precise spline modelling
- Fluent handling of complex models
- Relatively small size of saved files
- Wide palette of plug-ins allowing custom scripting

4.1.2 Grasshopper

This is a plug-in for Rhinoceros, which provides scripting in a very comprehensive, yet powerful way. It is launched in Rhino as a sub-window. It includes most of Rhino commands in form of components, which can be connected together. Resulting model is projected into Rhino environment in real time. This projection is, however, only virtual and unless generated ("baked", as it is called in Grasshopper), it cannot be used or modified directly in Rhino (Rutten, 2012).

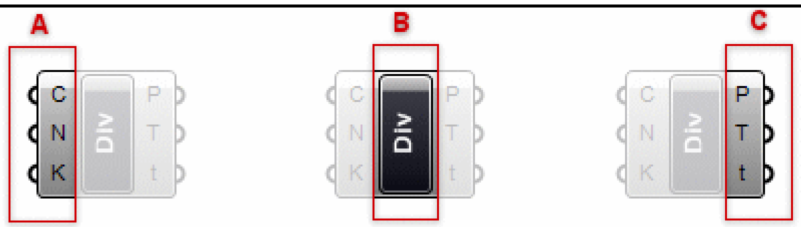


Figure 4-1: Grasshopper	Grasshopper component: Triple input pole (A), component tag (B), triple output pole (C)

4.2 Evolution

Many venues had to be discovered and many of them proved to be dead ends, before the final geometrical concept was established. This final one fulfils demands that were set in chapter previous chapters. In the text bellow one dead-end will be briefly outlined and then the final concept will be detailed.

4.2.1 "Flat Node" Concept

It was explored for about 3 weeks to finally prove no good prospects. It was node-governed. **Primary objective was a flat node**, which would be easy to cover by a watertight cap. Extrusions leading to such node would have to be deformed and twisted, so that they would meet in one plane – plane of the node.

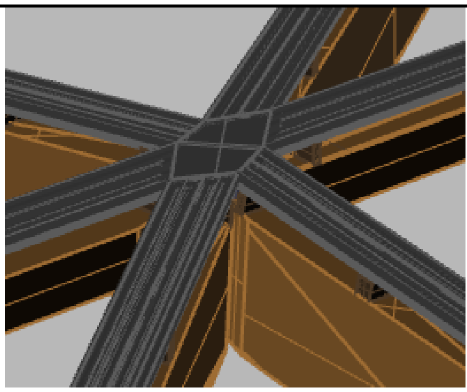


Figure 4-2: Flat node	Detail of a node of a gridshell structure (left). Non-continuous extrusion SLs are visible.

4.2.1.1 Advantages

Shaping of covering caps for nodes would be easy. As all the keder-SLs would intersect on one plane, it would be very easy to cover nodes by a watertight cap. In fact, it would only be a polygon respecting shape of the node.

4.2.1.2 Disadvantages

Definitely, the biggest disadvantage is to ensure intersection of keder SLs. Extrusions, on their way from node to node, would have to compensate difference in node-orientations. If some two system lines of primary structure were continuous through a node, system lines of corresponding extrusions would not. This, by itself, is enough to disable elegant parametrization so after many experiments a conclusion was made, that this concept would be abandoned. Its features and choices can be seen in the Annex IV.

4.2.2 Final concept of "Chamfering"

Having some experience with previous concept, I tried to start from another point – from the point of extrusions. Roughly, the procedure of this concept is as follows:

- Find middle surfaces of filling (either cushions or single-layers) so, that a gridwork of primary structure is offseted by a chosen distance
- Chamfer their edges such, that width of chamfered surface would be equal to desired extrusion width

4.2.2.1 Advantages

By definition, keder SLs must intersect, so the biggest problem of previous concept was easily solved right away.

Extrusions are straight and torsion-free in case of triangulated structures.

4.2.2.2 Disadvantages

Shape of the node becomes complicated, non-planar, thus the watertight capping of node becomes a more complex problem, whose solution will require some further scripting.

Chamfering length is individual for each edge. It means that those edges have to be identified and calculated separately.

4.2.2.3 Decision

Despite disadvantages, this concept was explored further and in the end proved to be worth it. In Figure 4-3 one can see how this concept handles above-mentioned choices; it must be applicable to any curvature and any basic polygon. Upstand length can be unique everywhere, although it is counted with only a few types (remainders to ideal shape have to be taken by elastic deformation of an extrusion – this is to be thoroughly calculated and implemented). Extrusions will be parallel to primary structure and twist will be handled both by pre-shaping in a shop (for large deformations) and forced deformation on site (for small deformations). Nodes will be irregular and covered either by a rubber pad and a flat cap (small non-planarity) or by a unique shaped piece (large non-planarity).

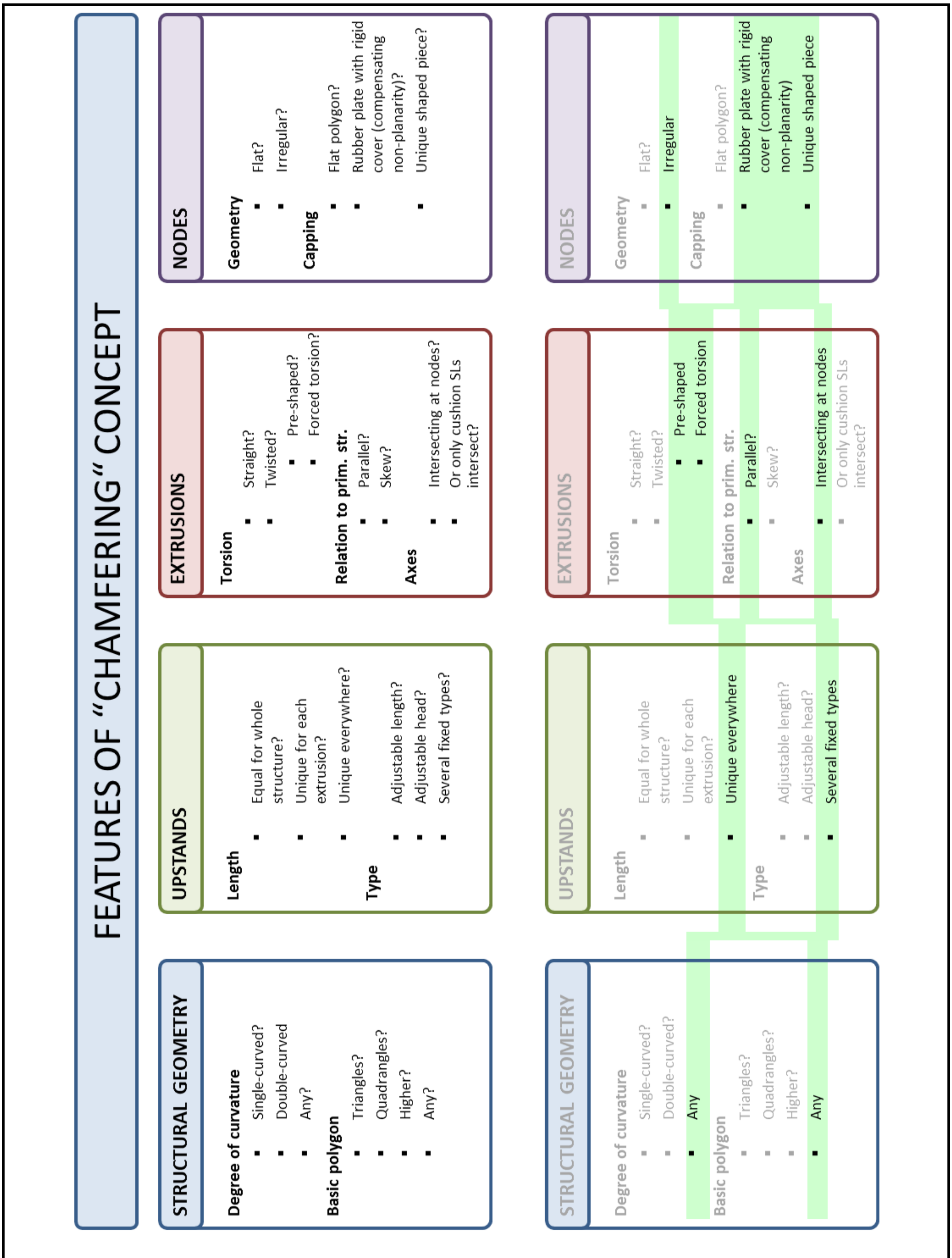


Figure 4-3: Features of "Chamfering"	Answers of "Chamfering" concept to given choices.

4.3 "Chamfering" Concept – Description of Process

Whole process follows a workflow as depicted in Figure 4-4.

4.3.1 Workflow

At the beginning, we have an input of a client. He has to provide geometry of primary structure (a 3D-model). From this we obtain **Axes of Primary Structure.**

Then again, client has to specify his wishes regarding average height of upstand, which influences overall appearance of resulting structure. This is a base for an engineer, who then has to verify possible collisions between primary structure and filling as well as to consider other potential demands coming from statics. From this we obtain **Height of Upstands.**

Based on filling type, choice of basic polygon of panelization and number of upstands per extrusion, an engineer has to make decision about a type of extrusion he will use. This determines **Extrusion Width.** This value stands for distance between two neighbouring filling-fields. "Real", physical, width of extrusion is slightly bigger. Value "Extrusion Width" is equal to distance of neighbouring "Keder SLs".

Now we have everything we need to start the **Parametric Script.** This script, created in Grasshopper plug-in, finds us all keder system-lines for given structure.

4.3.2 Tolerances

Resulting keder SLs have to be verified for precision. Following points have to be considered regarding our target-tolerance:

- Tolerance of extruding process of aluminium clamping profile
    - Usual tolerance of +/-0,8% of extrusion width (Engineers Edge, 2010)
- Compensation of cutting patterns
    - Common value of -0,5% in both dimensions – warp and weft (LEICHT, 2011). If designed exact value is distorted by other imprecisions, we can encounter a change in prestress followed by wrinkles or loss of stability of filling.
- Assembly precision on site – tolerance on location of members
    - Depends on point of view but for our purpose we consider it as much as 5mm.

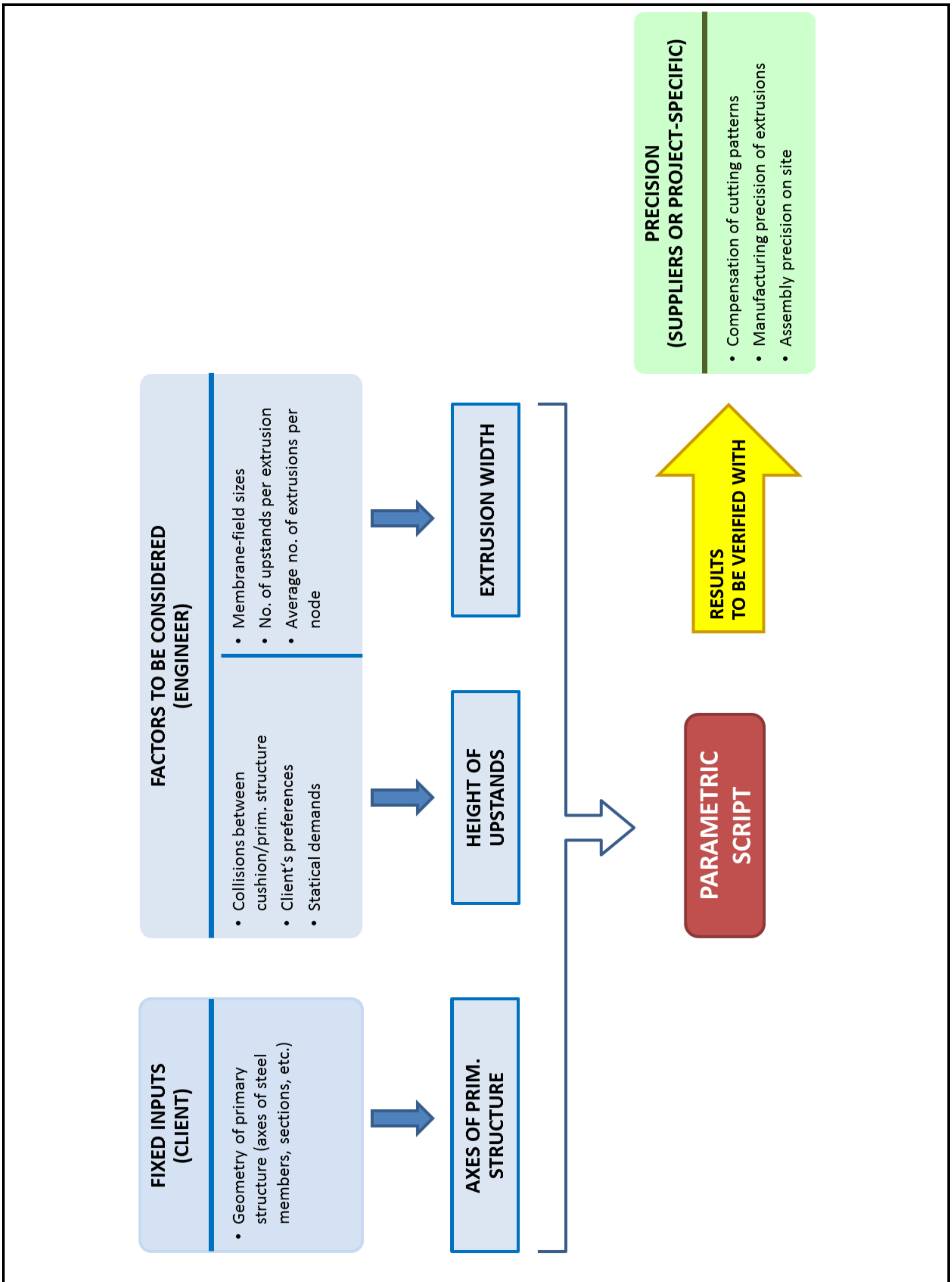


Figure 4-4: "Chamfering" concept workflow	Workflow diagram for final concept

4.3.3 Description of Chamfering Process

Chamfering length of an edge is a function of an angle between two surfaces adjacent to this edge. So before getting into the topic, let us introduce some variables with help of a scheme in Figure 4-5.

4.3.3.1 Theory

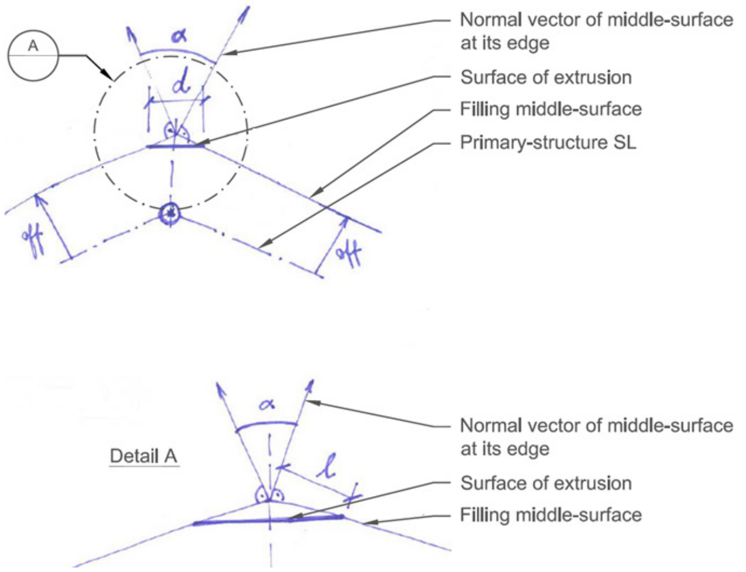


Figure 4-5: Chamfering Function	Variables, parameters and resulting values for chamfering function

Inputs:

- d = Extrusion width [mm]
- off = Offseting distance (see below, Step 2) [mm]
- α = Angle between two surfaces at their common edge [rad]

Resulting value:

- l = Chamfering length [mm]

A simple Equation 4-1 relates our parameters and variable to give us the resulting value:

$$l = \frac{d}{2 \cdot sin\left(\frac{\pi}{2} - \frac{\alpha}{2}\right)}$$

Equation 4-1: Chamfering length	Relation between parameters, variables and result

4.3.3.2 Manual Work Prior to Starting the Script

Prior to starting the script, we have to prepare all the input data. Principles of the parametric script will be illustrated on a real project.

4.3.3.2.1 Step 1: Offseting of primary-structure gridwork

System-lines of primary structure are then offseted by a chosen distance outwards. Offseting distance is equal to **Height of Upstands** from Figure 4-4. If the offseted gridwork needs some adjustments, it has to be done now.

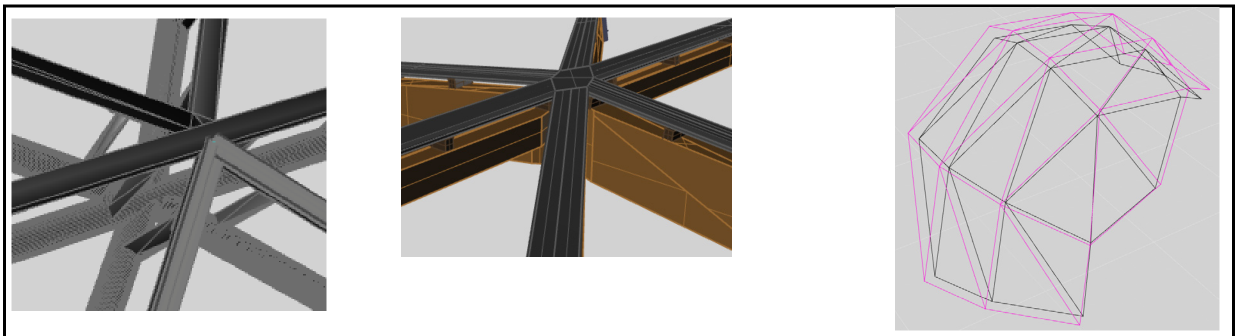


Figure 4-6: Step 1 Tall upstands, h=350mm (left); Low upstands, h=75mm (middle); System lines of primary structure in BLACK, offseted system lines in PINK (right)

4.3.3.2.2 Step 2: Creating of middle-surfaces

Within offseted system lines we now need to represent middle-surfaces of filling. If triangle is chosen as a basic structural polygon, then middle-surfaces can be drawn directly in Rhino; middle-surfaces are planar. While if the basic polygon has more than 3 vertices, preliminary formfinding should be done and resulting surfaces then imported back to Rhino.

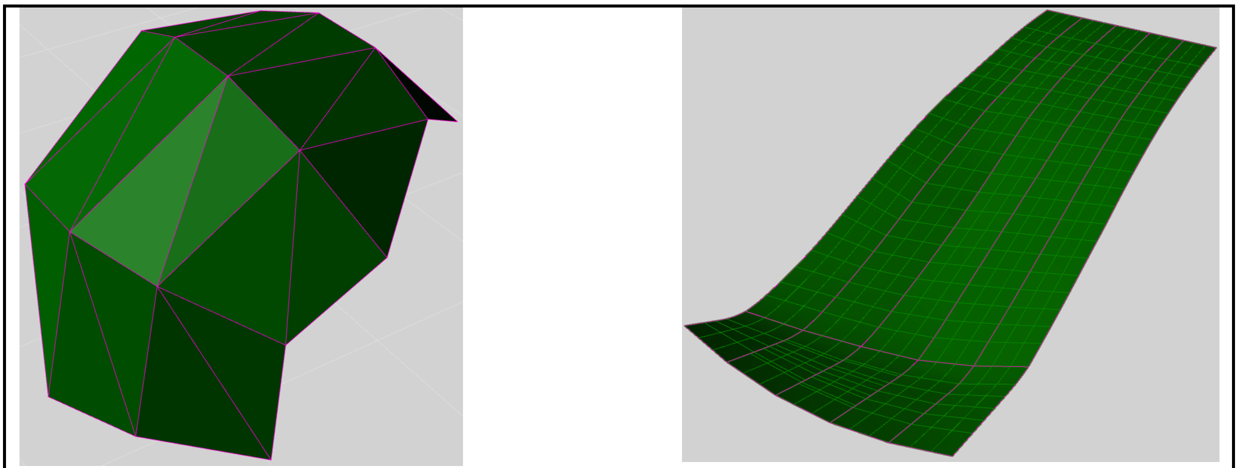


Figure 4-7: Step 2 Offseted system lines (PINK) and generated middle-surfaces (GREEN). Surfaces generated directly on triangulated structure (left). Surfaces generated through formfinding (right).

4.4 Illustration by a Real Project

To illustrate principles of the parametric script, let's introduce it to a real project. It is an offer for a roof of a shopping mall in Singapore which is quadrangulated cushion structure with considerably curved clamping profiles; a serious challenge to quality of the script and a good opportunity to discover and fix its weak points.

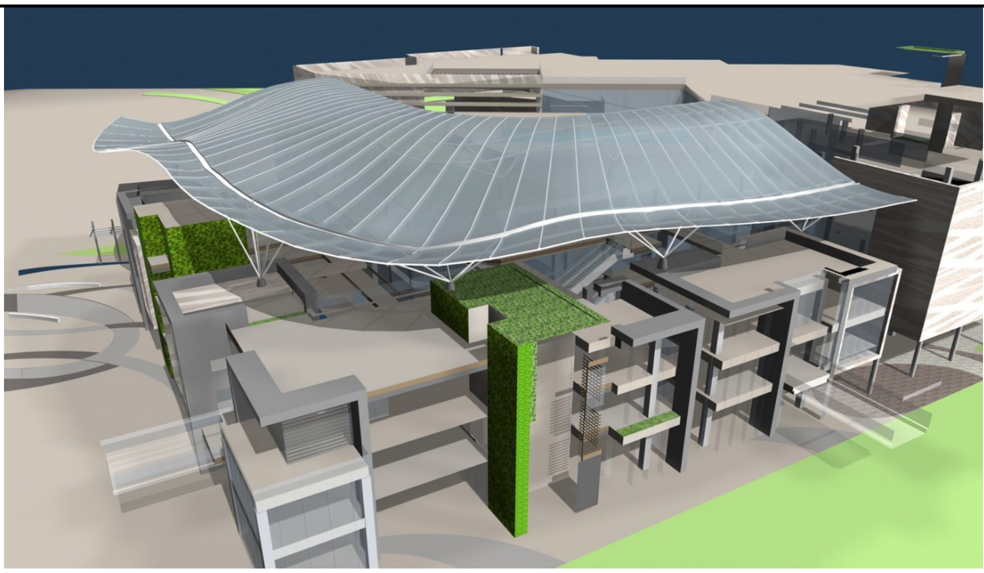


Figure 4-8: Global view	Overall view of the projected roofing. Dimensions of big, upper, cushions is app. 40x3m, Dimensions of small, lower, cushions is app. 8x3,5m. Overall area covered with cushions is about 4500m^2.

Because middle-surfaces were already provided by client, we were able to skip the manual work and we could directly feed the parametric script by these surfaces.

4.4.1 Functional Principles of the Parametric Script

Fully detailed operation can be consulted in Annex IV, whereas code itself from Grasshopper is shown in Annex 8.1.1.10.

Script works as a **loop**. In each **step** of the loop it does following operations:

- Pick'n'Find
- Calculate chamfering length
- Generate keder SLs
- Check extrusion width

4.4.2 Pick'n'Find

In this sub-step, the script picks one surface (0^{th} in this case), and finds all its neighbours (in red). This is necessary as it provides information on which surfaces should be compared in calculation of chamfering length.

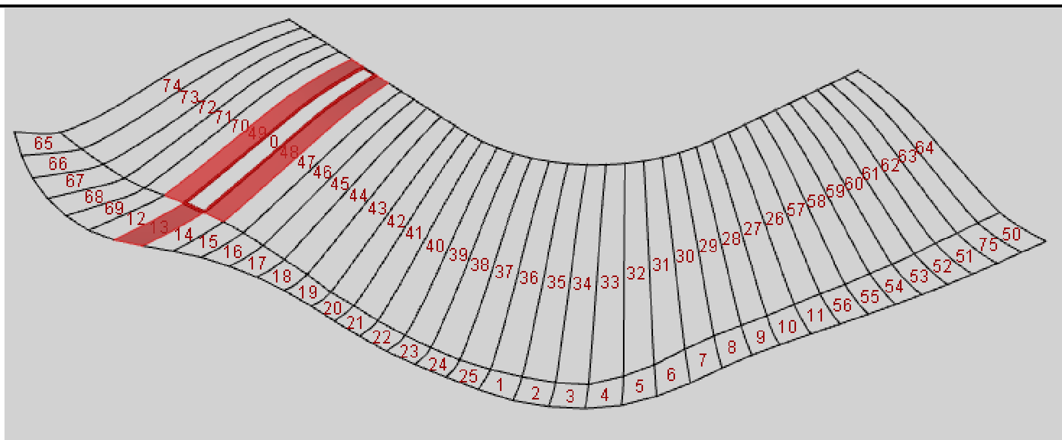


Figure 4-9: Pick'n'Find	Surface #0 picked, and its neighbours (marked in red)

4.4.3 Calculate Chamfering Length

Having marked appropriate surfaces, the script now identifies their common edges. Then, respecting Equation 4-1, it determines the chamfering length at both ends of each edge.

In case where the basic polygon has more than 3 vertices, the chamfering length will not be constant along whole edge. Instead, it is interpolated between two exactly calculated values at the ends (further explained in Annex IV).

In either case chamfering length determines radius of a cylindrical surface that is swept along the common edge.

4.4.4 Generate keder-SLs

Now, having the cylindrical surface and "Picked'n'Found" surfaces, it makes an intersection of all of them, while filtering out all duplicates and buds. Like this we obtain our keder-SLs. The resulting set of curves is then sorted in couples, where each couple represents one extrusion (or clamping profile).

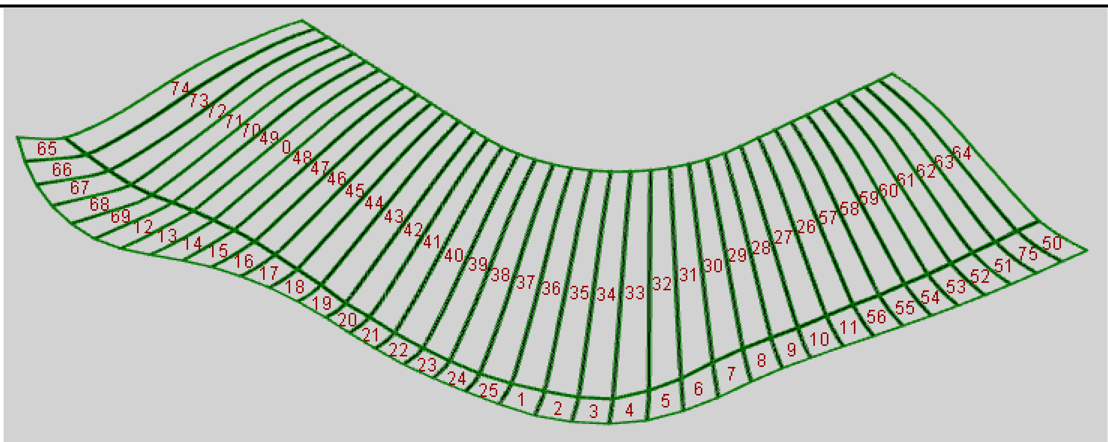


Figure 4-10: Keder-SLs	After all surfaces were picked and duplicate SLs were filtered, we have our final cushion SLs.

4.4.5 Check Extrusion Width

Distance between keder SLs in each couple has to be equal to value **"Width of Extrusion"** that was set as an input. This is important for quadrangulated structures, where interpolation is involved in calculation of chamfering length.

All **PROPERLY GENERATED** keder SLs fit within tolerance of 0,5%. Compared to tolerance of fabrication of aluminium extrusions (0,8%, see paragraph 4.3.2 Tolerances), this value is more than sufficient.

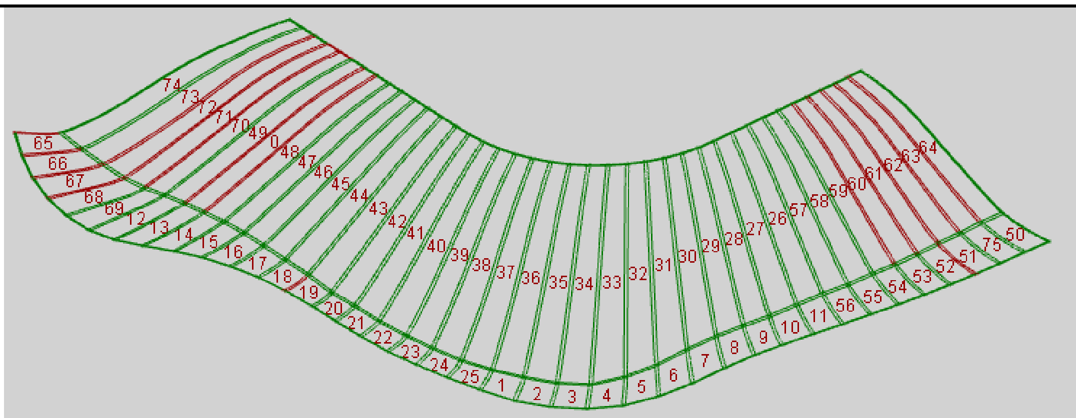


Figure 4-11: Filtered SLs	Properly generated keder-SLs within tolerance of +-0,5% of extrusion-width (green) and those not-properly generated (red)

As already hinted, there were keder-SLs that were **NOT PROPERLY GENERATED**. This defect involves integrity of Grasshopper itself. When problematic steps were undertaken manually, in Rhinoceros, there were no such problem. For more details see Annex IV.

4.5 Problem of Upstands

Problems of upstands stayed with no exact parametric solution due to lack of time. However, based on experience from previous parametric scripting, it is believed that adding an appropriate section to the existing parametric script will not cause major difficulties.

Presumed workflow follows:

- Select number of upstands per meter
- Divide extrusions accordingly
- On the primary-structure axis, find a closest point to division points of extrusions
- Generate upstand-SLs

4.6 Conclusion to GEOMETRIC Problem

Although finding a good concept of solution showed to be exceptionally time-consuming, final concept proved to be robust and accurate.

There is still work to do, but the biggest questions are answered and main direction is now given. Finishing this problem is not a matter of invention anymore, but rather a matter of time.

Despite some bugs and a missing part (upstands), high precision of generated geometry was approved to be sufficient for professional use.

5 INTERFACE PROBLEM

With all the geometry defined, next step is to find a way how to **TAKE** the geometry, **IMPORT** it in a FEM software and **CALCULATE** and **OPTIMIZE** it.

5.1 Methods

It was decided to take Martin Brown's NDN (Brown, 2011) as a FEM-software of choice.

5.1.1 NDN software

It is a new FEM-software, currently under development. Its author is Martin Brown, a recognized expert in lightweight structures (FabriTec Structures).

As one can see in the Table 5-1, each node has its ID number (column "Node"). While beams are defined by two points (I-node and J-node), these point-IDs are used to identify them as well (see Table 5-2).

```
NODE DATA: 184
 Node  Typ  Lyr  Blk  DoF   X-Coord   Y-Coord   Z-Coord
    1    0    0    0    2   10.0000   10.0000    0.0498
    2    0    0    0    2    9.7781   10.0000    0.1589
```

Table 5-1: NDN file – points Sample of a NDN source file – format of points

```
ELEMENT DATA: 180 Beams
 Elem  Typ  Lyr  Blk    I    J    K  Mat Name
    1   20    0    0    1    2    0    0
    2   20    0    0    2    3    0    0
```

Table 5-2: NDN file – beams Sample of a NDN source file – format of beams

5.1.1.1 Advantages

- Calculation of membrane-elements through Minimal Surface method. This is in general case more pregnant method than Force Density Method as noted in § 3.1.4.2.
- Includes supporting structure and its stiffness into membrane calculation.
- Possibility to design supporting steel structure according to American, Australian and British codes (Brown, 2011). Support for other codes under development.
- ASCII format of files. It is easy to feed any data inside through a text-editor (see a sample in Table 5-1).

5.1.1.2 Disadvantages

- Software under development by a single developer, therefore a lot of minor bugs.
- Load cases can't be user-defined.
- Can't handle NURBS. Only linear segments and circular arcs.
- Others to be discovered later…

5.1.2 Microsoft Excel

It is used as intermediary between Grasshopper and NDN. The reason is easy manipulation with data organized in rows and columns, so that it can be saved as a text file – a source for NDN.

5.2 Solution

In this part we will discuss a proposed solution.

5.2.1 Outline

Previously obtained geometric data has to be prepared for further processing. Since NDN can't handle NURBS, the geometry had to be discretized. This, again, was done by means of a parametric script done in Grasshopper. See Figure 5-1 for a workflow that was applied for data preparation. As inputs we use filtered keder SLs from chapter 4, while setting roughness of discretization (number of segments per meter)

As a result, we will be able to extract:

- Centre-points, by which extrusion-axes run through
- All connections (beams) between centre-points, while omitting redundant ones (to assure that end of one extrusion will NOT be connected to beginning of another)
- Torsion rates of each extrusion from one centre-point to another
- Nodes, by which keder SLs run through (to define filling panels)

After completion of the preparation phase, we will be able to import all this data to an Excel spreadsheet, save it as a text file and, eventually, import the whole geometry in NDN. There the FEM-analysis can begin, starting with formfinding.

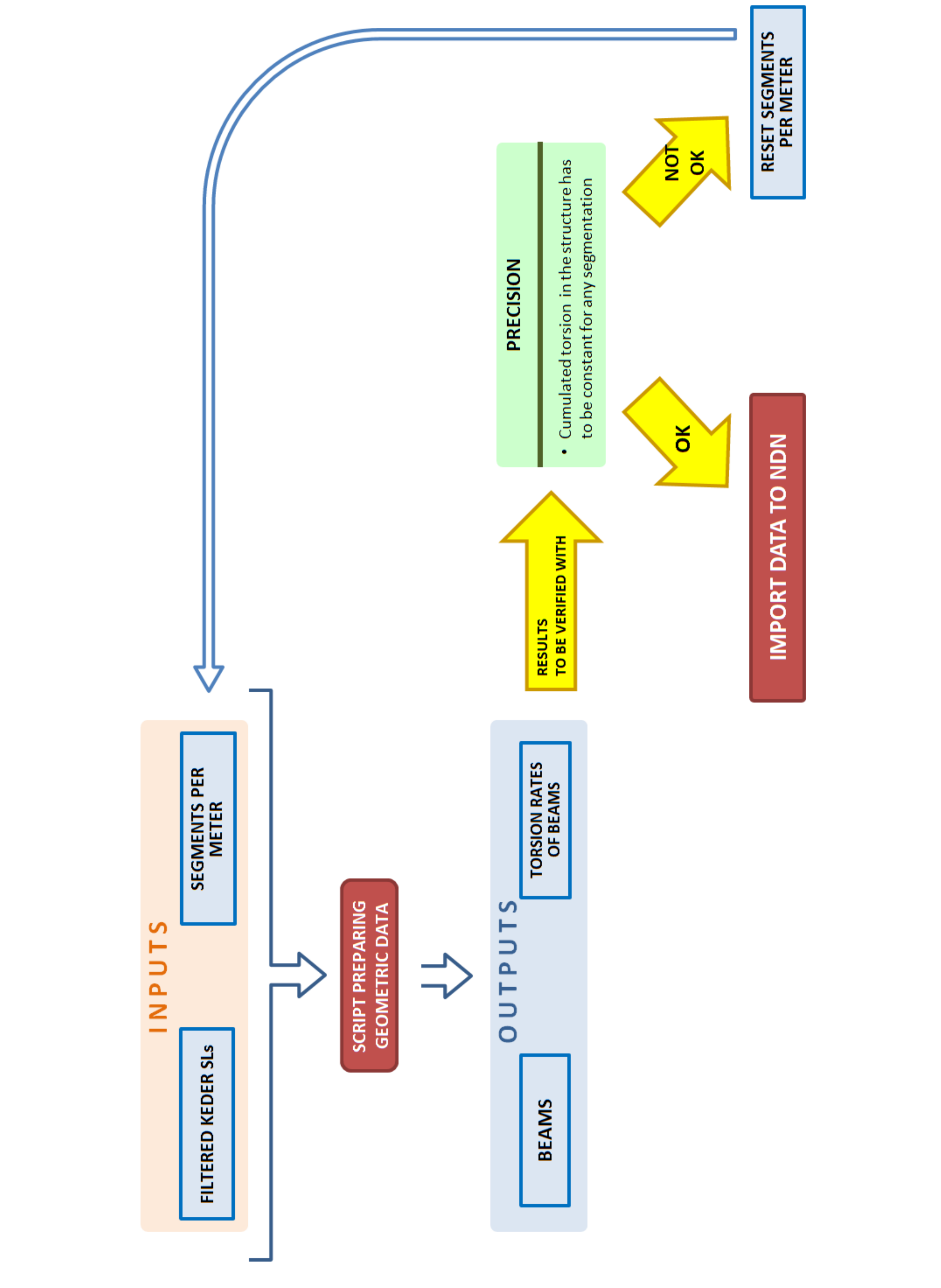


Figure 5-1: Workflow of data-preparation	Diagram showing workflow of data-preparation. If prepared data fulfils precision demands, it is forwarded to import to NDN.

5.2.2 Principles of the Parametric Script

In following steps, let us explain how the parametric script works. Verifying algorithm was included as well. Again, the "Singapore Project" was used as a reference. As it was found out, **NDN can handle only limited number of nodes**. That is why only a smaller segment was chosen. Reprint of the script can be found in Annex VII.

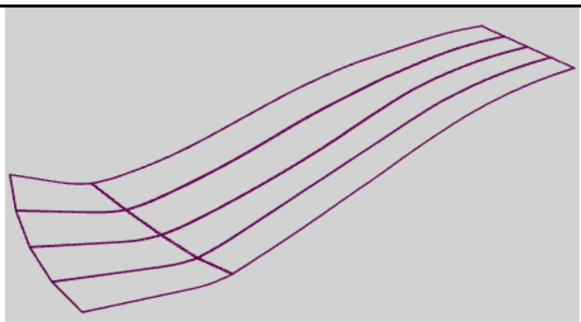


Figure 5-2: Segment used as ref.	Segment of "Singapore Project" which was used as reference geometry to develop and illustrate INTERFACE problem.

5.2.2.1 Inputs

From previous work, we feed keder SLs organized in pairs. We also make a decision of a number of segments per meter length of extrusion.

5.2.2.2 Split Curves into Segments

Accordingly to inputs, ONE curve of each couple is split by division points. Subsequently, the script creates a link connecting each division point with closest point on the SECOND curve. Like this, it is sure, that these links between each two points will be perpendicular to both original curves. The closest points are then used to split the SECOND curve.

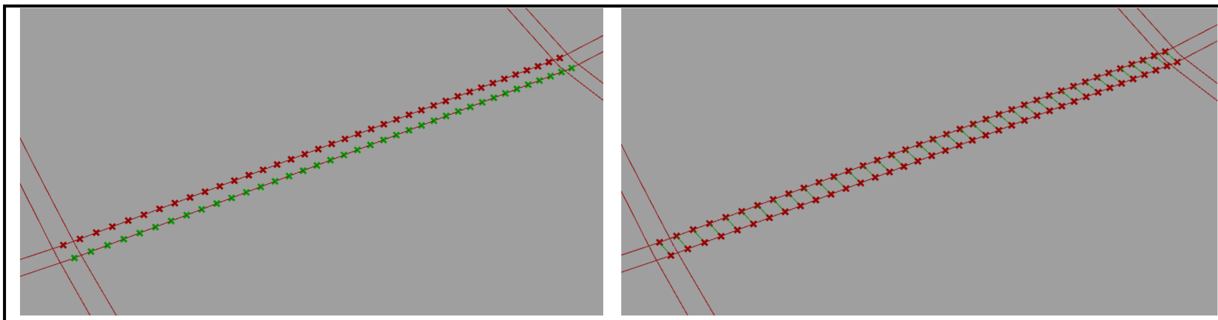


Figure 5-3: Split curves	Division points (red) and closest points (green) (LEFT) Links connecting each division point to its closest point (green) (RIGHT)

5.2.2.3 Generate Axes of Extrusions

Extrusion axes are, by definition, leading through centre-points of links.

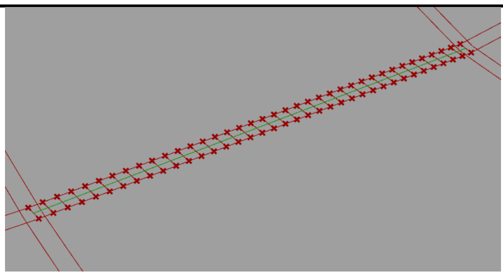


Figure 5-4: Axes of extrusions	Axis of an extrusion (green) led through centre-points of links

5.2.2.4 Measure Torsion Rates

Its principles are explained in following points and illustrated in the Figure 5-5:

1) Pick Nth link by its centre-point and move it to a centre-point of (N+1)th link
2) Project Nth link to a plane given by tangent vector of extrusion axis and (N+1)th link
3) Measure angle between projected Nth link and (N+1)th link. Measured value is the **torsion rate** between Nth and (N+1)th segments

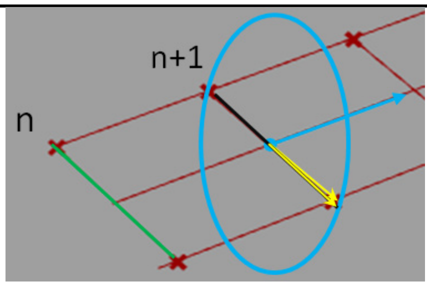


Figure 5-5: Torsion rates	Nth link (green), (N+1)th link (black), plane given by a tangent vector of extrusion axis and (N+1)th link (blue), vector of projected Nth link and (N+1)th link (yellow)

5.2.2.5 Cumulated Torsion Test

Keeping in mind, that by discretization we lower the precision, a check had to be done to evaluate this loss.

Objective of this is to find a minimum for "segments per meter", so that the calculation is as fast as possible, while maximum precision is maintained. It is project-specific value, thus it is relevant only to the one project, from which it was derived. This value is important during optimisation, when several iterations are gone through.

The test is based on summing up of all torsion rates in whole structure. This value we call "cumulated torsion".

We set a wide range of values for "segments per meter". For each value, the script calculates "cumulated torsion" of whole structure. Then the results are evaluated on a statistical basis.

Absolute deviation (AD) and Median absolute deviation (MAD) was chosen as a best statistical approach for its robustness – it doesn't get disturbed by outliers (David C. Hoaglin, 1983). Limiting values are project-specific, depending on phase of project as well as its character

$$\mathrm{MAD} = \mathrm{median}_i\left(\left| X_i - \mathrm{median}_j (X_j) \right| \right)$$

Equation 5-1	Median absolute deviation

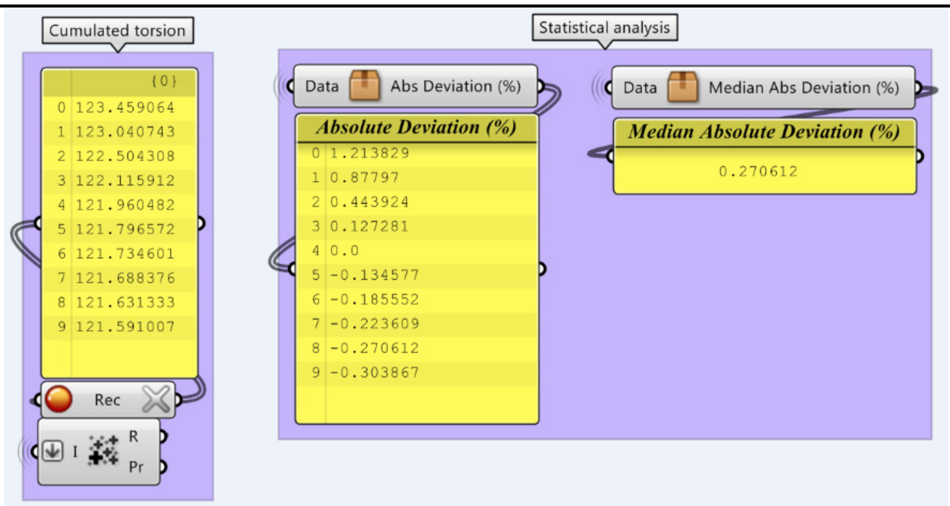


Figure 5-6: Statistical evaluation	"Segments per meter" set as a series from 1 to 10.

Although this test helps us to determine maximal acceptable roughness from the point of view of torsion rates, it does not deal with directional and positional accuracy of extrusion-axes. This has to be looked at in further stages of development. Proposed solution to it could be **adaptive roughness of discretization** – with increasing local curvature, the optimal number of "segments per meter" would grow. Very curved portions would be discretized more accurately, while straight portions would respect the optimal number.

5.2.3 Importing Prepared Data into the NDN

Having all the necessary data prepared, it is not too complicated now to export all the data to Excel spreadsheet and to import it to the NDN. All we need is a direct exporter component in Grasshopper (scripted in Visual Basic (Crawford, 2010)) and a template *.xls file formatted in such a way, that the NDN can read it.

Two template files were created – one for nodes, the other for beams. Appropriate data is then imported into them and the two are eventually merged. Result is saved as *.txt file, closed, and the extension is re-written as *.ndn.

5.2.4 Adjusting and Exploiting of Imported Model in the NDN

After a successful import, we have to assign section and material properties to lines. As long as we have a full template for beams, this can be in future done automatically, by the parametric script. For now, it is done manually. In the Figure 5-7 you can see a portion of a Singapore project imported already in NDN.

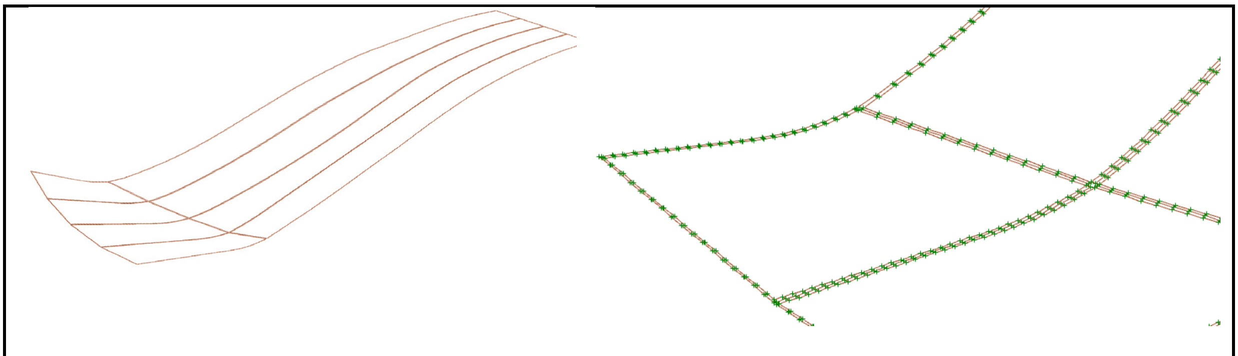


Figure 5-7: A segment imported in the NDN	A global view of successfully imported structure (left); a close-up of a corner with highlighted nodes in green (right)

5.2.4.1 Assigning of Section and Material Properties

Properties are assigned as follows:

- **Axes of extrusions:** section properties of a real clamping profile, mass of real aluminium.
- **Links:** infinite stiffness, zero weight
- **Keder SLs:** infinite stiffness, zero weight

This is due to fact, that a beam element can't be represented by two axes. So the realistic properties are assigned only to extrusion-axes, while links and keder SLs are there only to provide proper linear support to filling panels.

In the Figure 5-8 you can see a detailed view on an extrusion with assigned properties.

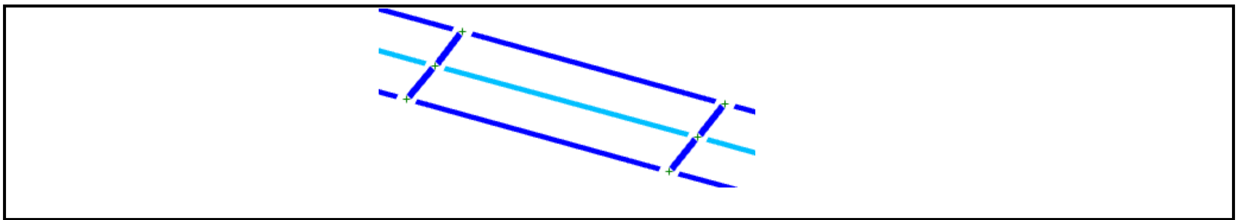


Figure 5-8: **Detail on extrusion in the NDN**	Realistic section and material properties for extr.-axes (light blue) Zero weight, infinite stiffness for links and keder SLs (dark blue)

5.2.4.2 Further Exploitation of the Model

Afterwards, it was planned to explore the torsion data and, in function of its magnitude, to set it either as a load case (imposed deformation) or to set it as fixed geometry. Eventually, filling-panels should have been fitted within keder SLs. However, at this point some other imperfections of NDN showed up:

- It is impossible to set imposed torsional deformation. Instead it is replaced by "jacks", whose use is imprecise and over-complicated and close-to-impossible to verify in a large scale. It would have been necessary to put two jacks at each free node of extrusion-beams. This implies tens of thousands of jacks.
- It is impossible to define a membrane-border by a segmented line of random shape. This, in fact, dismisses NDN from service for such structures.

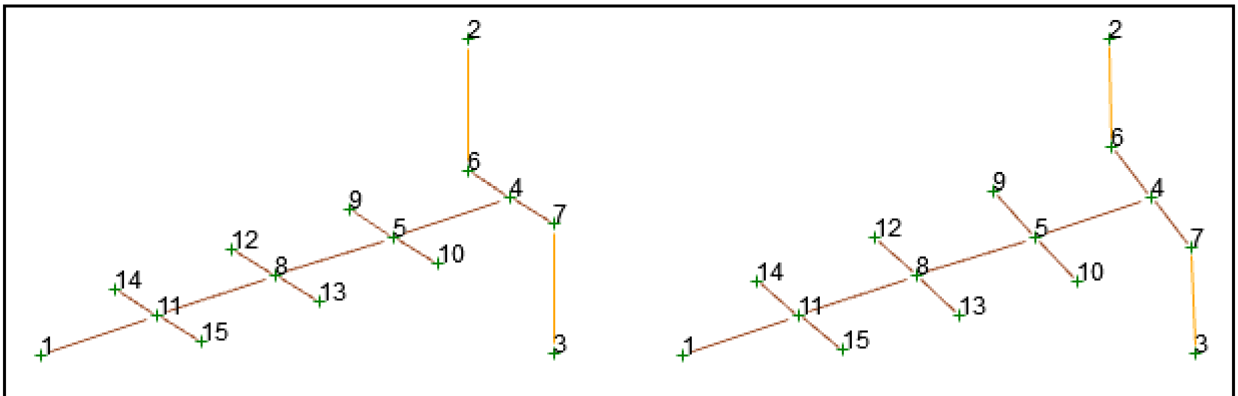


Figure 5-9: **Use of "Jacks"**	Original shape (LEFT); Deformed shape after jacks were actuated (RIGHT) Brown – beam elements; Yellow – jack elements; Supported nodes – 1 and 4.

5.3 Conclusion to INTERFACE Problem

Work done in the topic of INTERFACE is clearly outlined and used methods are sufficiently robust and ready for trial professional use. However, some work is still left to do in order to produce a really smooth, transparent and user-friendly interface.

Experience showed, that NDN software is not yet in a state where it can be used for typical projects done at LEICHT and only future will tell, if the developer was able to work fast enough to implement all the missing key-features.

6 STATICS PROBLEM

Due to time consumption of two previous chapters GEOMETRY and INTERFACE, there was no time left to really start with some calculations. In following paragraphs I will outline a proposed approach, which I would have followed provided there was time.

6.1 Objectives

Main objective was already mentioned in paragraph "3.4.3 Problem of Clamping Profiles (Extrusions)" on page 33. That is: How much of torsional deformation can a clamping profile accommodate during assembly on site, so that it can still withstand all designed loads? And will the workers on site be able to overcome resulting forces necessary to bring a clamping profile to its designed shape?

Equilibrium of these two problems is a key factor influencing economy of production of clamping profiles. If they are given too much work at the shop, the manufacturing costs more money, while installation is too comfortable. On the other hand, if they will be used till the ULS, it might be very difficult for workers on site to install them and cost of work would explode. "Equilibrium" is, when installation is still reasonably fast and comfortable, while shaping in shop is minimized.

6.2 Proposed Approach

In following points let us break it down. A workflow diagram can be seen in Figure 6-1.

1) Leave all torsion data as input geometry for calculation.
2) Load the structure with all designed combinations and evaluate reserves in resistance (ULS) and in deflection (SLS).
3) For extrusions, where the reserves were big enough, we change the torsion data **from input** geometry **to imposed** deformation (generating stresses in the section). Evaluate necessary assembly forces. If too high, then
4) Restart the calculation and check the reserves again.
5) Keep optimizing the structure.

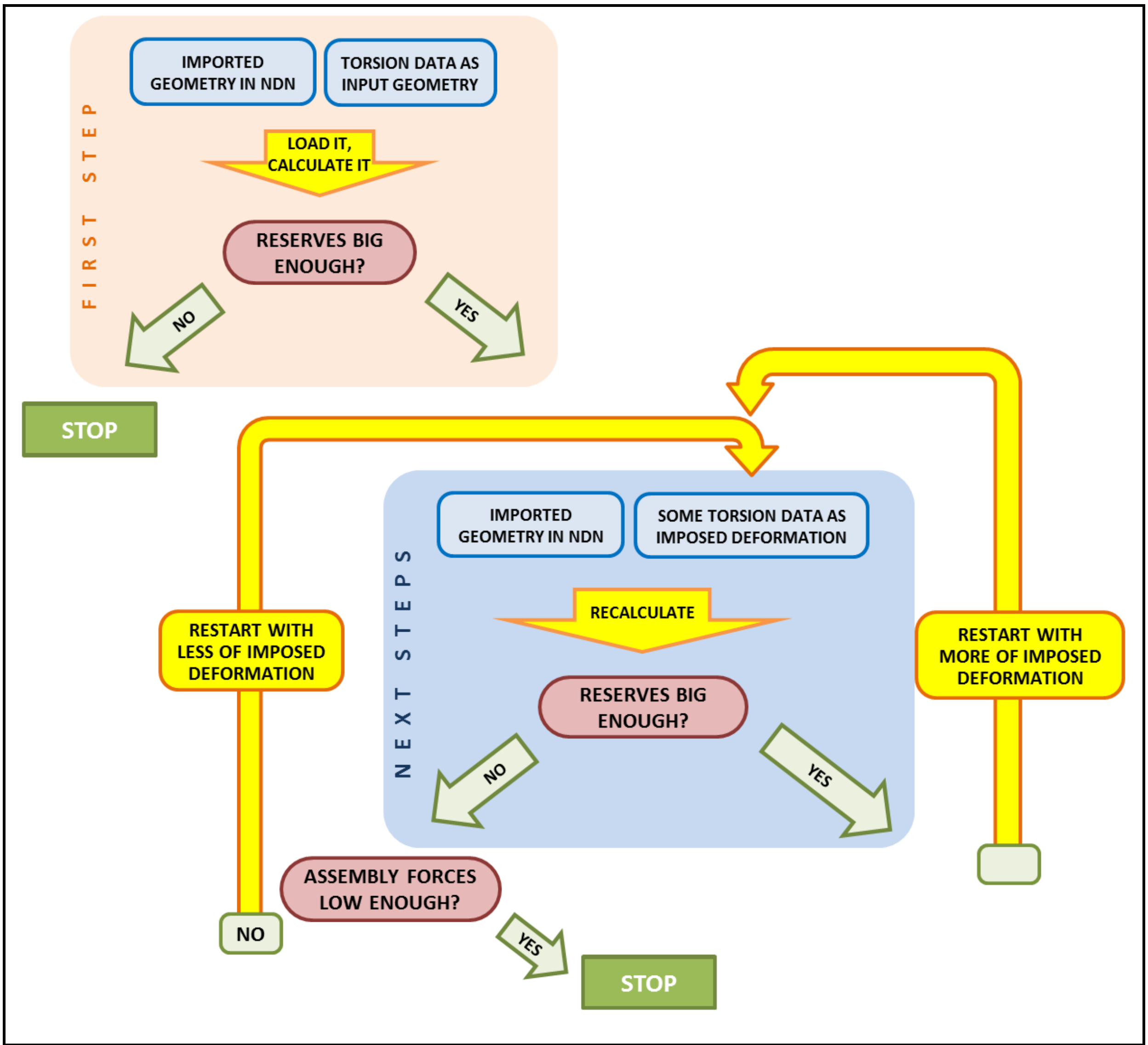


Figure 6-1: Workflow of STATICS	Illustration to a proposed workflow of chapter STATICS

6.3 Conclusion to STATICS Problem

Proposed approach has to be explored and a working solution has to be found in order to bring the tool to its full operational state. It is not out of question, that some other FEM-software will have to be used to replace NDN.

7 FURTHER WORK

7.1 GEOMETRIC Problem

It is mainly reliable generation of keder system-lines that has to be fine-tuned. And as mentioned in 4.6, problem of upstands has to be dealt with in a complex way, according to proposed outline.

7.2 INTERFACE Problem

Section and material properties could be set parametrically, by Grasshopper. The way how to do it is known, though the decision mechanism must be specified.

Whole INTERFACE part would be considerably boosted, if it were able not only to WRITE a text file, but also to READ it, so that it could react and, for instance, adjust sections.

It might be that NDN software will be replaced, if – despite perfect support from Martin Brown – development doesn't prove to be fast enough.

7.3 STATICS Problem

This is still an open problem with just some propositions based on experience with both NDN and INTERFACE. Its solution is essential in order to fulfil the ambition of a complex tool.

In the best scenario, this part would be completely subordinate to seamless INTERFACE, so that all the project-related decisions (regarding statics of clamping profiles and upstands) would be done on one place.

8 CONCLUSION

In general, an outline to all problems was given and all major question and ambiguities were answered. However, should the objective be a complete tool for handling clamping profiles and upstands, there is still much work to do in all three domains.

However, main goal from the Introduction was achieved. Crucial questions were answered and functioning framework was set. Namely in the question of GEOMETRY and INTERFACE, the work is already well advanced and application proved to be sufficiently robust. This, by itself, means an acquisition to host company, indeed.

Having signed a working contract with the host company, the author will be most probably given time and space to further improve what he started by this work.

Bibliography

Brown, Martin. 2011. membrane NDN Software. [Online] NDN Software, 1 January 2011. [Cited: 20 April 2012.] http://www.ndnsoftware.com/.

Crawford, Scot. 2010. Grasshopper to Excel Exporter. [Online] 19 April 2010. [Cited: 30 April 2012.] http://lmnts.lmnarchitects.com/featured/grasshopper-to-excel-exporter/.

David C. Hoaglin, Frederick Mosteller, John W. Tukey. 1983. *Understanding Robust and Exploratory Data Analysis.* s.l. : John Wiley & Sons, 1983. ISBN 0-471-09777-2.

Demidov, Evgeny. 2010. *An Interactive Introduction to Splines.* [Website] s.l. : Ibiblio, 2010.

Engineers Edge. 2010. Engineers Edge. [Online] 20 January 2010. [Cited: 10 June 2012.] http://www.engineersedge.com/extrusion.htm.

FabriTec Structures. [Online] [Cited: 25 March 2012.] http://www.fabritecstructures.com/#/utilizing-the-expertise-of-dedicated-and-experienced-professionals.

LEICHT, Structural engineering and specialist consulting. 2011. *Advancia Centre.* Paris : LEICHT Structural engineering and specialist consulting GmbH, 2011.

LEICHT, Structural engineering and specialist consulting GmbH. 2011. *Formfindung und Zuschnitte mit EASY.* [Document] Munich : s.n., 2011.

—. 2007. *Internal ressources.* [Pictures] Rosenheim : s.n., 2007.

—. 2008. *Summary of biaxial testing.* [Internal document] Munich : s.n., 2008.

Moritz, Karsten. 2007. *ETFE film as a structural element.* Technical University Munich, Munich : 2007.

Otto, Frei. 2009. ILEK. [Online] University of Stuttgart, March 2009. [Cited: 1 June 2012.] http://www.uni-stuttgart.de/ilek/1/homepage/.

Robert McNeel and associates. 2010. Rhinoceros, NURBS modeling for Windows. [Online] 1 January 2010. [Cited: 22 February 2012.] http://www.rhino3d.com/.

Rutten, David. 2012. Grasshopper, Generative modeling for Rhino. [Online] Robert McNeel and Associates, 1 January 2012. [Cited: 22 February 2012.] http://www.grasshopper3d.com/.

Sederberg, Thomas W. 2010. *Computer Aided Geometric Design Course Notes.* [Document] Provo, Utah, USA : Bringham Young University, 2010.

Weininger, Florian. 2010. *Lecture notes on freeform structures.* [Lecture notes] Munich : s.n., 2010.

Annexes

I. Illustration to Chapter "3.1.1 Classification"

Figure 8-1: Dif. curvature	Single-curved (left) and double-curved (right) surfaces ET = tangent plane; P=common point; t=tangent line

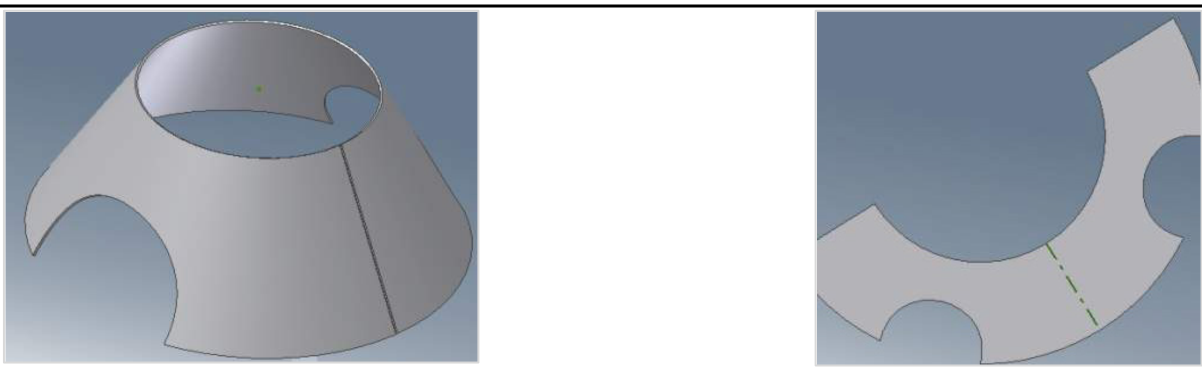


Figure 8-2: Developing	Single-curved surface (left) and its developed shape (right)

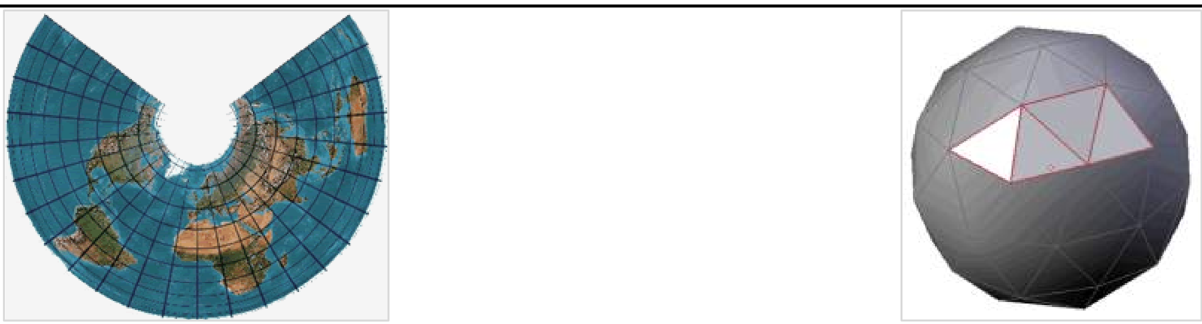


Figure 8-3: Developing	Double-curved surface in its projected (left) and discretized (right) shape

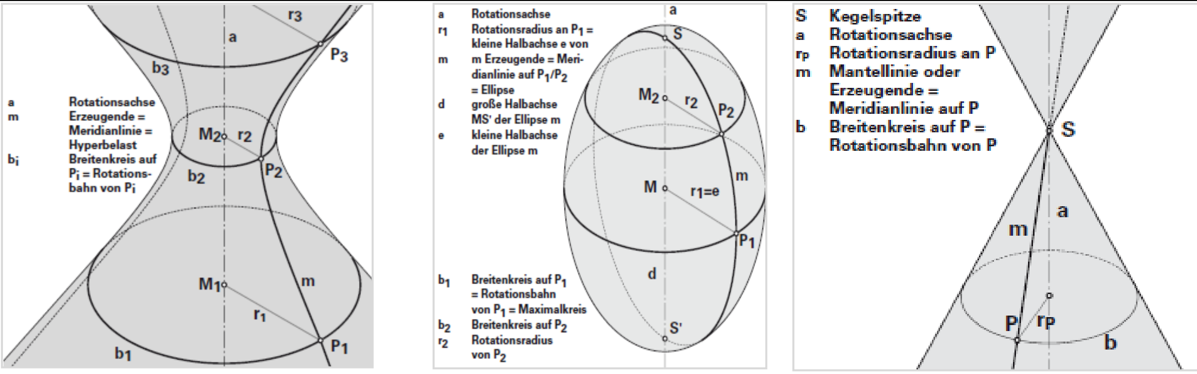


Figure 8-4: Srf of Revolution	Surfaces of revolution. From left: Hyperboloid, Ellipsoid, Cone

Figure 8-5: Srf of Revolution	Application of surfaces of revolution in architecture. From left: Hyperboloid (Shukhov Tower, Nizhny Novgorod, Russia, 1896); Ellipsoid (James Law's Cybertecture, Mumbai, project); Cone (Cone Tower, Melbourne, Australia, 2005)

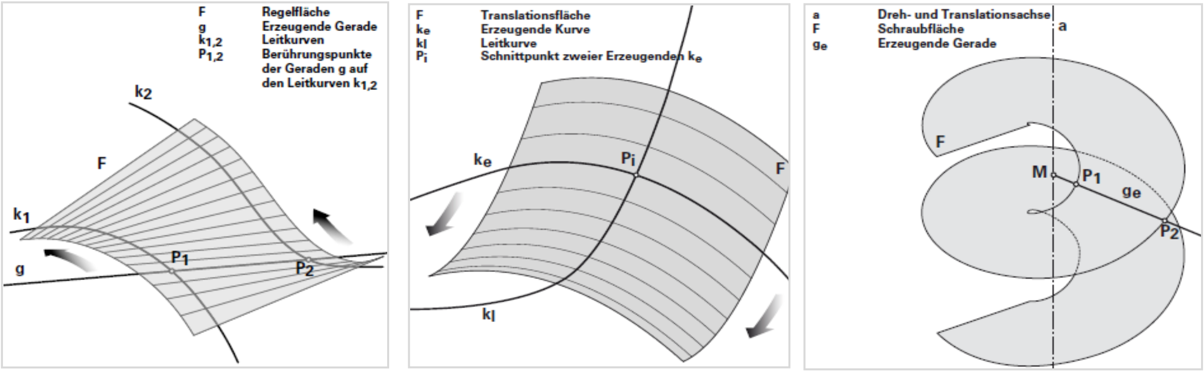


Figure 8-6: Translation srf	Various translation surfaces generated by a generatrix "g" along a rail "k"

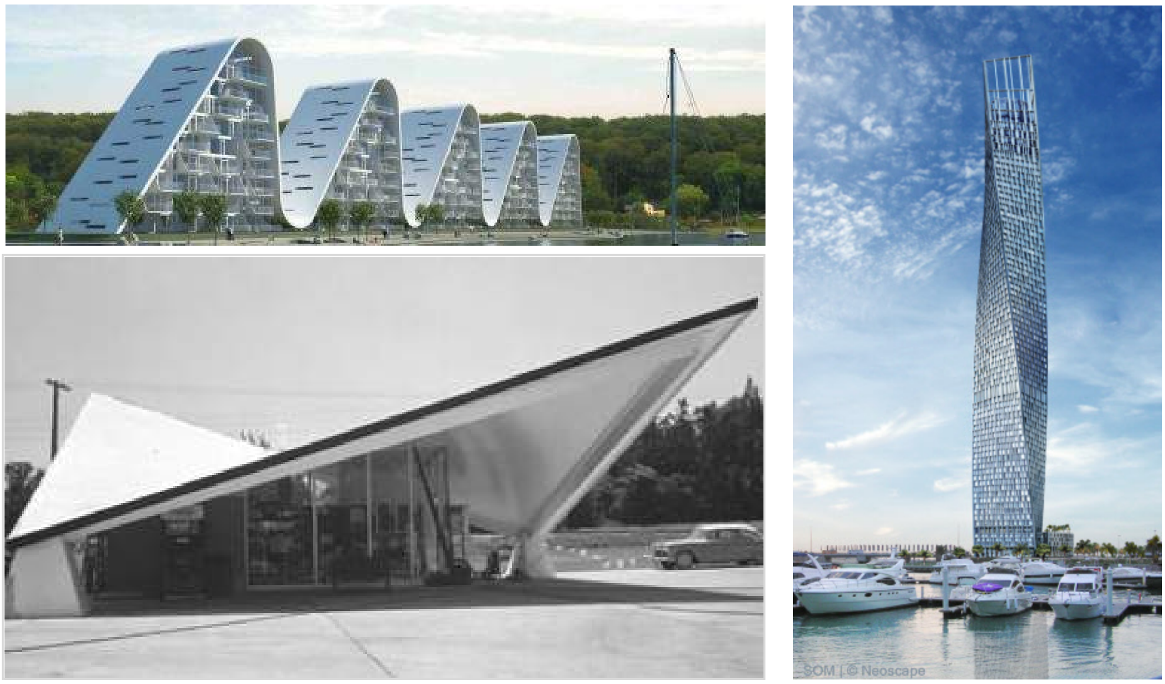


Figure 8-7: Translation srf	Application of translation surfaces in architecture. The Wave in Vejle, Vejle, Denmark, 2009 (above left); Cardwell Service Station, Cardwell, Australia, 1958 (below left); Infinity Tower, Dubai, UAE, under construction (right)

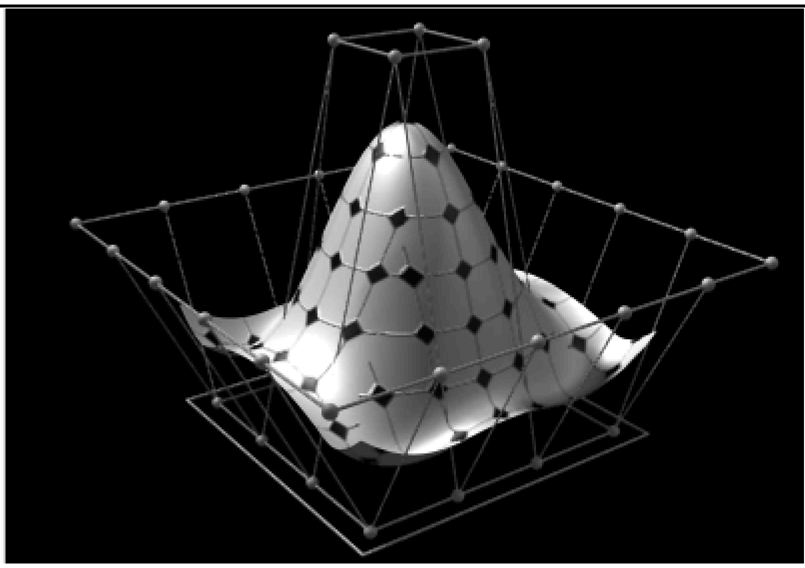


Figure 8-8: NURBS srf	NURBS-Surface given by its control points, or control polygons.

Figure 8-9: NURBS srf	Application of NURBS-surfaces in architecture. Nuovo Polo Fiera, Milan, Italy, 2005

II. Forms of Membrane Structures

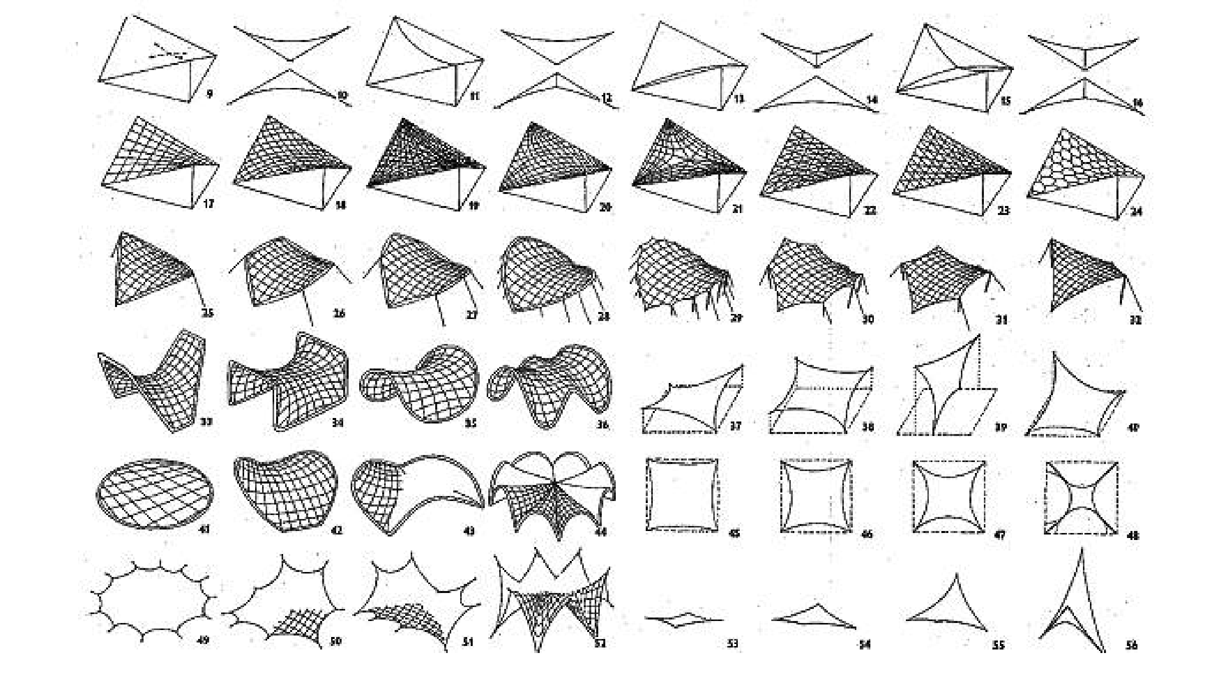


Figure 8-10: Membrane shapes	Saddle forms

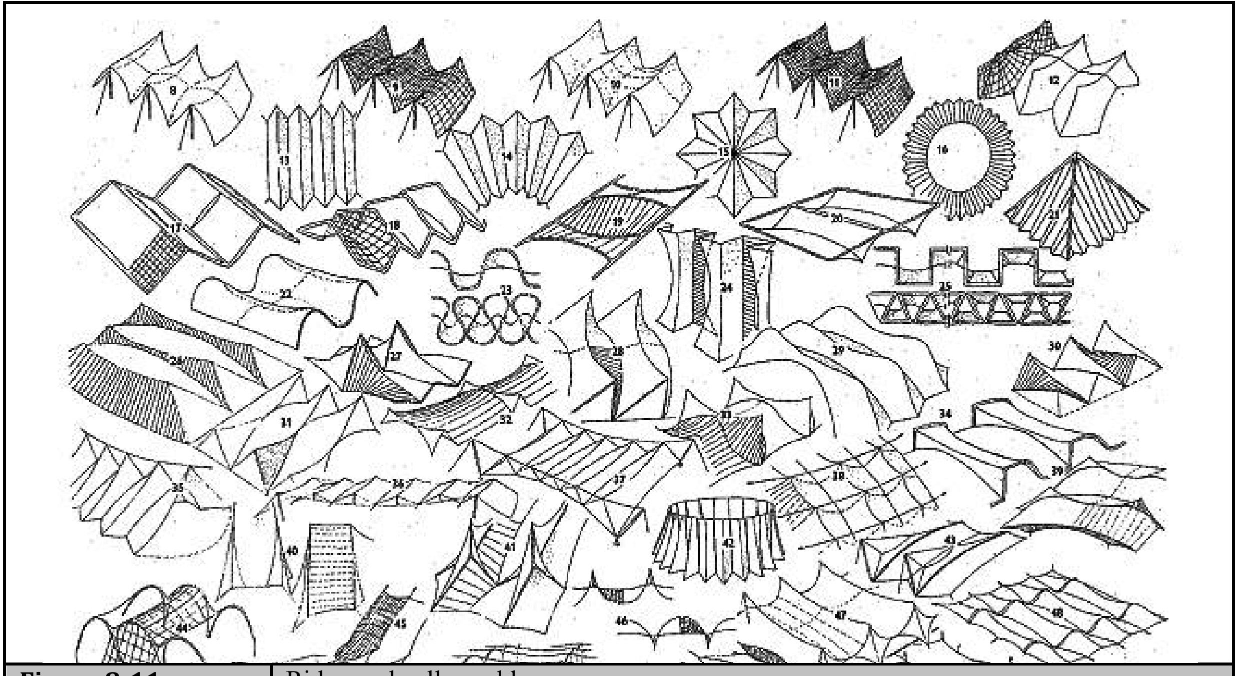


Figure 8-11:	Ridge and valley cables

Membrane shapes

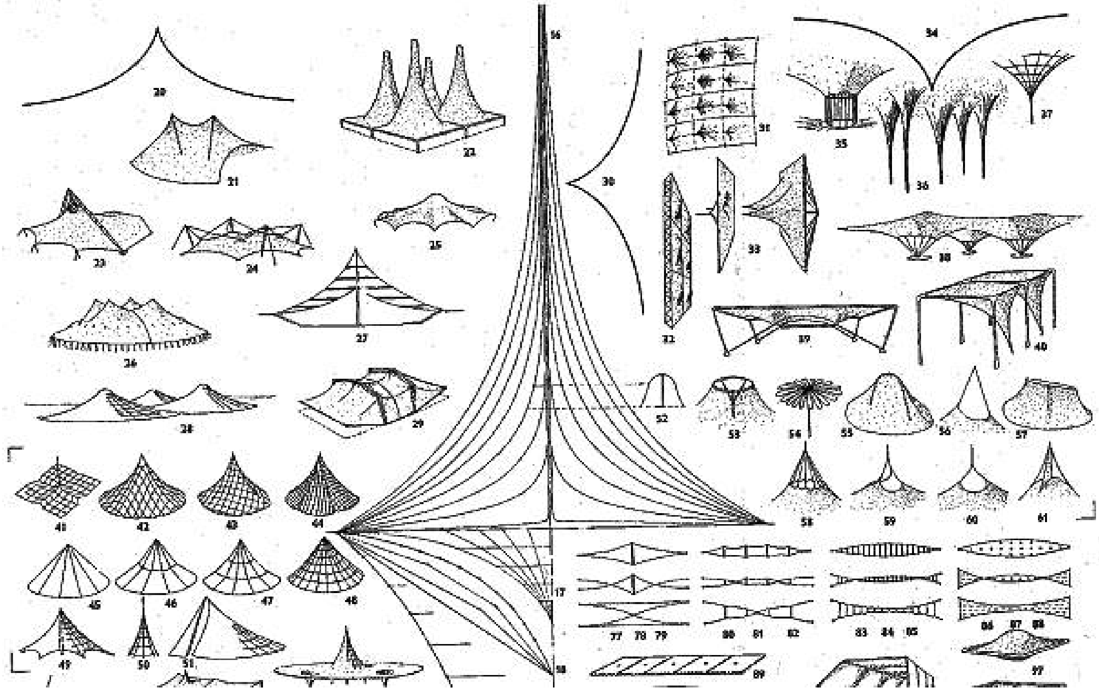


Figure 8-12: Membrane shapes High points, deep points

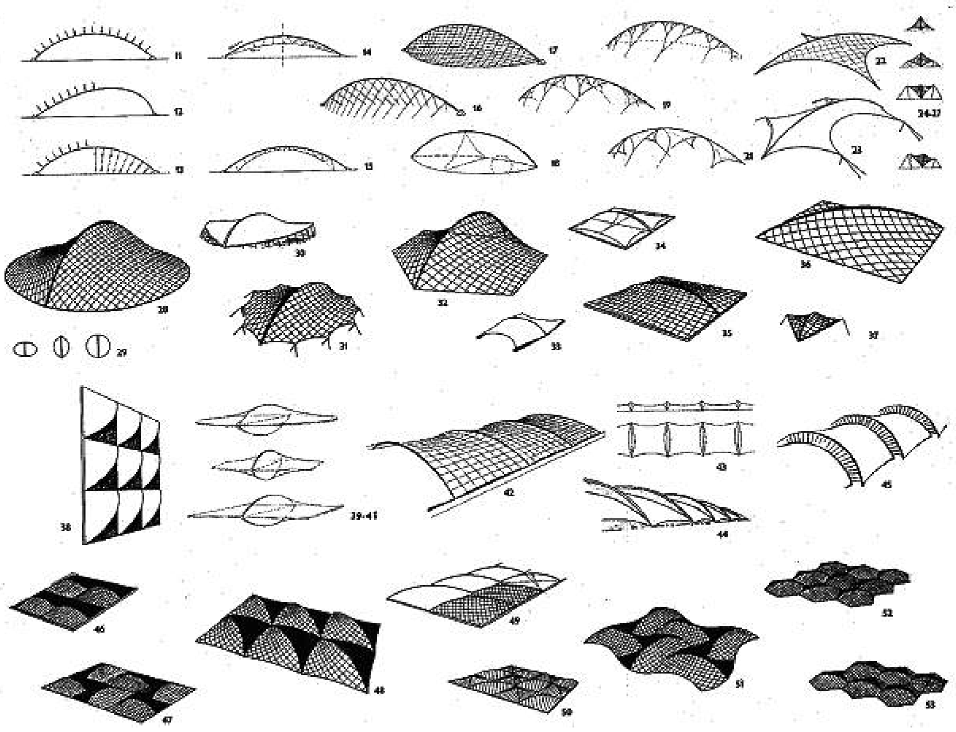


Figure 8-13: Membrane shapes Linear supports

III. Material details for membrane structures

Gewebematerial (ohne Gittergewebe)	Materialtyp	Flächengewicht [g/m²] nach DIN 55 352	Mindestwerte der Zugfestigkeit Gewebe Kette/Schuss [N/5 cm] nach DIN 53 354	Bruch- dehnung Gewebe [%] Kette/Schuss nach DIN 53 354	Weiterreißfestigkeit Gewebe [N] Kett/Schuss nach DIN 53 363
Baumwollgewebe		350 -520	1700/1000 2500/2000	35/18 38/20	60 80
PTFE-Gewebe		300 520 710	2390/2210 3290/3370 4470/4510	11/10 11/1018/9	ca 500/500
ETFE-Gewebe THV-beschichtet		250	1200/1200		
Polyestergewebe PVC-beschichtet	Typ I Typ II Typ III Typ IV Typ V	800 900 1050 1300 1450	3000/3000 4400/3950 5750/5100 7450/6400 9800/8300	15/20 15/20 15/25 15/30 20/30	350/310 bis 1800/1600 580/520 800/850 1400/1100 1800/1600
Glasfasergewebe PTFE-beschichtet		800 1150 1550	3500/3500 5800/5800 7500/7500	7/10 bis 2/17	300/300 bis 500/500
Glasfasergewebe Silikonbeschichtet		800 1270	3500/3500 6600/6000	7/10 bis 2/17	300 570
Aramidfasergewebe, PVC-beschichtet		900 2020	7000/9000 24500/24500	5/6	700 4450

Table 8-1: Membrane properties Typical characteristics of membrane materials

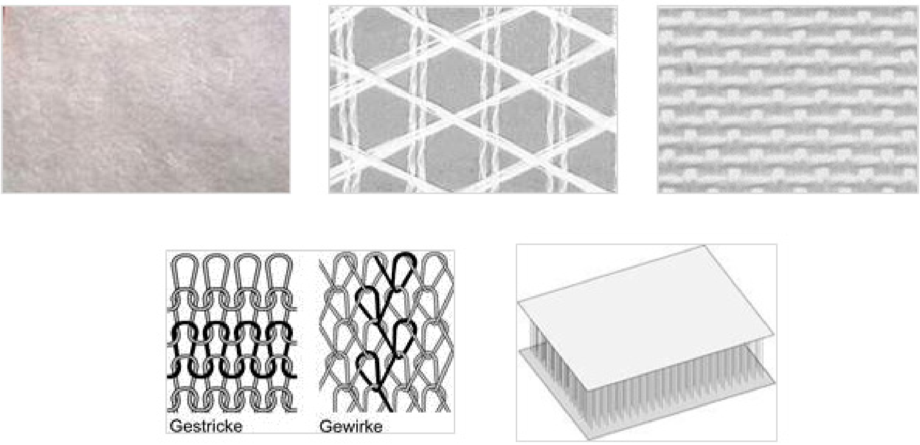


Figure 8-14: Membrane structures Possible types of load-bearing membrane materials (from left to right, from up to down): Vlies (non-woven), Multi-directional tape, Woven, Knitted, 3D-woven

LEICHT
Structural engineering and specialist consulting GmbH

Anhaltswerte für die Materialeigenschaften von ETFE-Folien

Dichte			1,75	g/cm³
Mechanische Werte bei 1-achsiger Beanspruchung und bezogen auf 23 °C	Bruchspannungen		52,5	N/mm²
	Spannungen bei 10 % Dehnung		21	N/mm²
	Bruchdehnung		600	%
	Kriechdehnung	bei 6 N/mm²	min. 1	%
	Temperaturausdehnungskoeffizient		10-13	10^{-5} / K
Ingenieurkonstanten bei 1-achsiger Beanspruchung und bezogen auf 23 °C	Querdehnzahl		0,45	
	Elastizitätsmodul		300-750	N/mm²
Strahlung Referenzwerte für 200 µm	Transparenz t-Wert	klar: 0,92	bedruckt, silber, 50%: 0,59	
	Transmissionsgrad – UV-Bereich	klar: 0,71	bedruckt, silber, 50%: 0,48	
	Gesamtenergiedurchlaßgrad g-Wert	klar: 0,89	bedruckt, silber, 50%: 0,63	
Brandschutz-klassifizierungen	Deutschland	DIN 4102	B1	Nach veralteter Normung mangels aktueller Prüfergebnisse
	Großbritannien	BS 476 part 6&7	class 0	

Figure 8-15: Membrane properties | Sample technical sheet of ETFE foil used at company LEICHT.

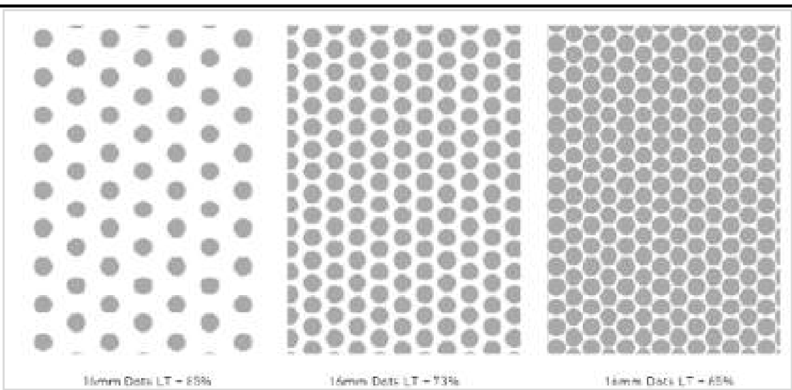


Figure 8-16: Membrane textures | Different kinds of pattern, giving different translucency.

IV. Features of "Flat Node" Concept

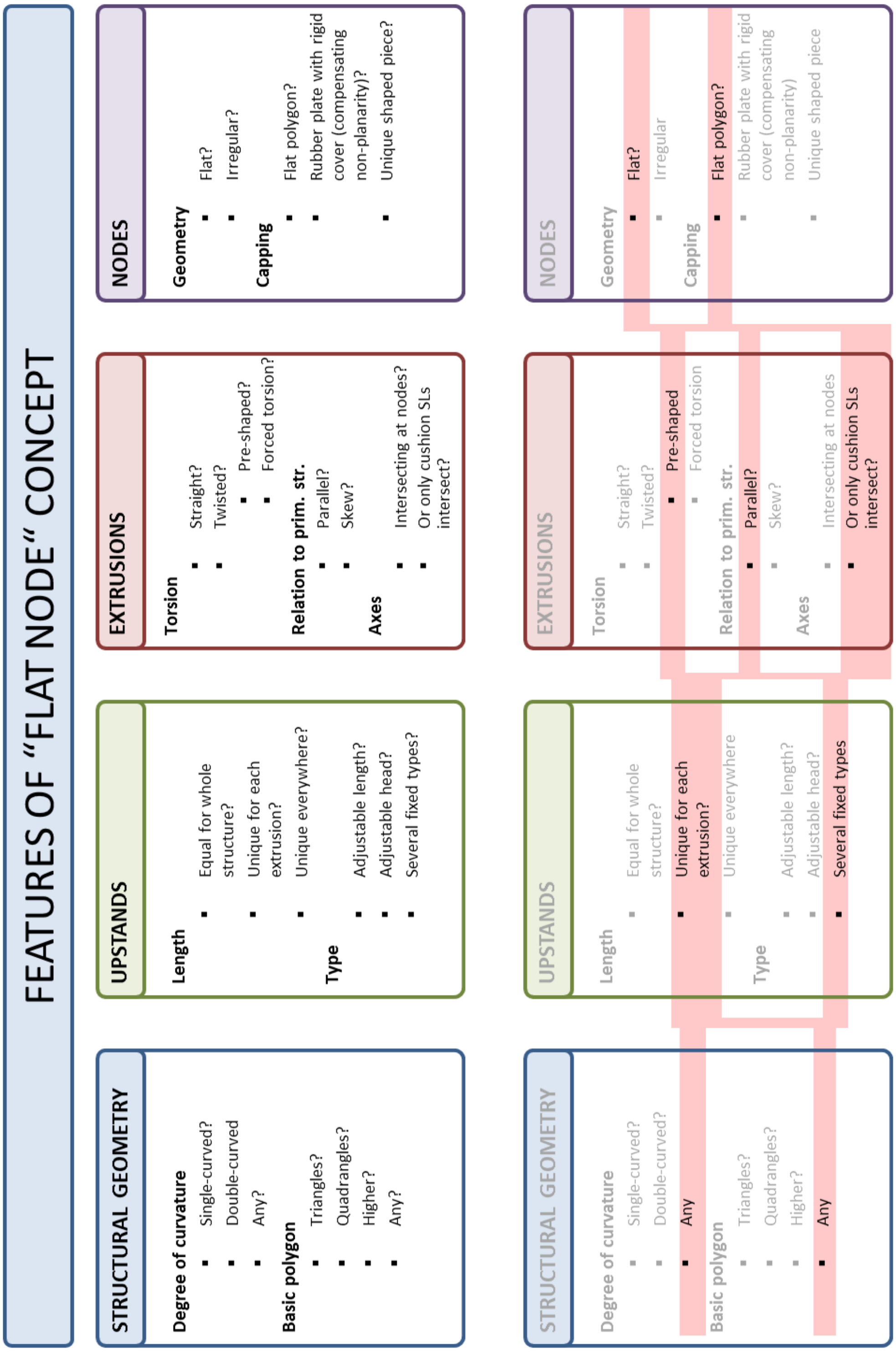


Figure 8-17: Features of "Flat Node"	Features and choices made for "Flat Node" concept. This one was later abandoned as it proved incapable of solve the geometry properly.

V. Detailed Operation of the Parametric Script "Chamfering"

Chamfering process itself is a loop following this diagram:

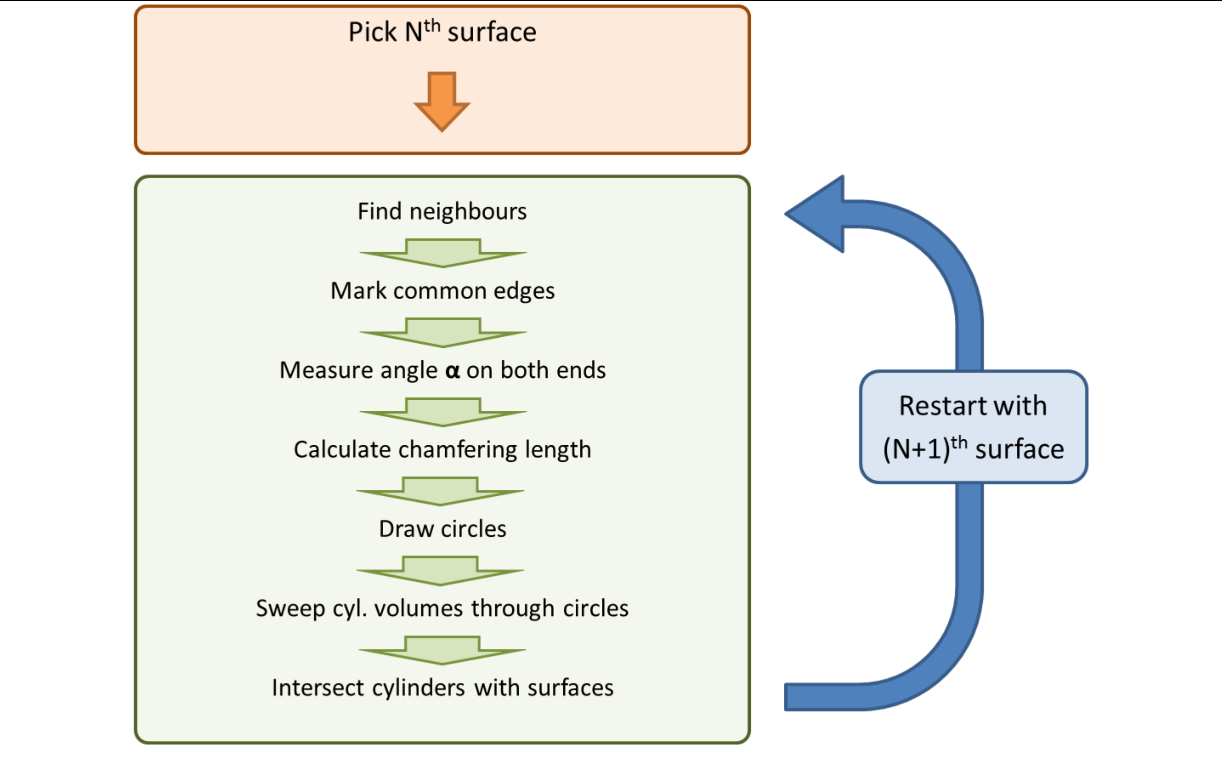


Figure 8-18: Script diagram	Algorithm diagram of the parametric script

8.1.1.1 Pick Nth Surface

Typically, sequence for N starts with 0 and runs through a series of numbers, until there are no surfaces left to compute. See Figure 8-19.

8.1.1.2 Find Neighbours

In this step, the script finds all neighbouring surfaces to one that is picked, based on distance between edges (if the distance is zero, then this surface is neighbour). See Figure 8-19.

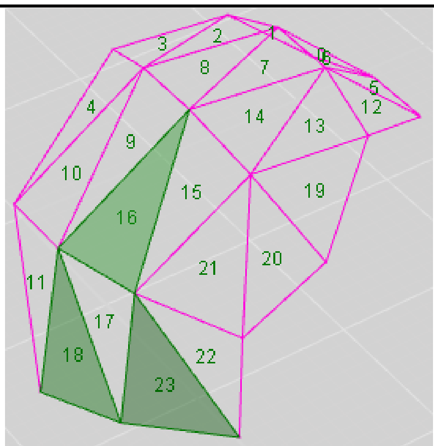


Figure 8-19: Pick'n'Find Pick Nth surface (N = 17 in this case), Find its neighbours (no. 16, 18 and 23)

8.1.1.3 Mark Common Edges

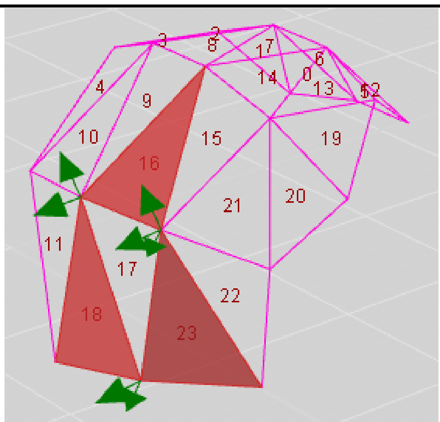
Aim is to create couples of picked and neighbouring surfaces. For instance, if basic polygon is triangle, than we end up with three couples, where each neighbouring surface appears only in one couple and the picked one appears in each one.

8.1.1.4 Measure Angle α on Both Ends

Now we have to measure angle α on both ends of each edge. Reminder: angle α is the angle between normal vector of neighbouring surfaces at their common edge. See Figure 8-20.

8.1.1.5 Calculate Chamfering Length

We can calculate chamfering length for each angle α based on chamfering function (see Equation 4-1).



	{0}
0	215.947395
1	215.947395
	{1}
0	203.417351
1	203.417351
	{2}
0	200.094603
1	200.094603

Figure 8-20: Angle α Normal vectors of surfaces at common edges (left); Chamfering lengths for both ends of each edge (three sets per two items). Because the basic polygon is triangle and surfaces are planar, calculated chamfering length is equal on both ends

If the basic polygon of structure is triangle, it means that middle-surfaces are PLANAR, than chamfering length will be constant through whole length of the edge. Analogically, if the basic polygon has more than 3 vertices, angle α may not be equal on both ends of an edge (see figure below). Obviously, chamfering length for whole edge is then **linear interpolation between its exact extreme values**. Application to a real quadrangulated structure (see paragraph 4.4, page 42) has shown maximum deviation of 0,5% of the extrusion width. Given tolerances of assembly and fabrication, this value is more than sufficient.

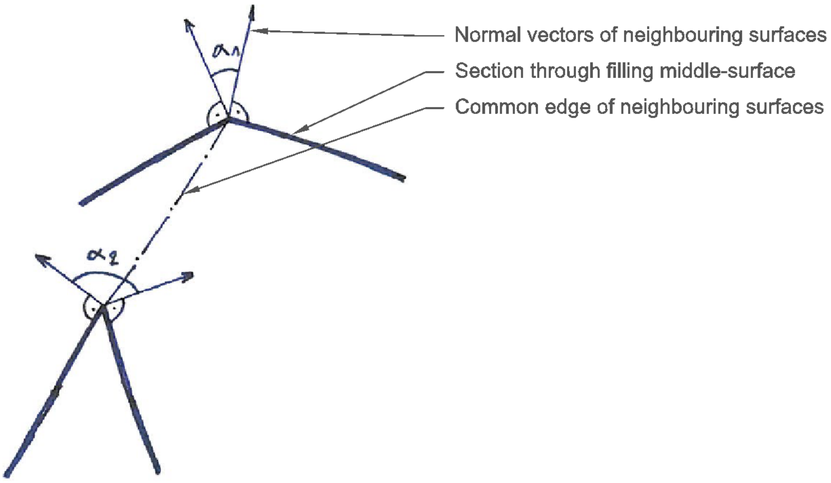


Figure 8-21: Quads If basic polygon has more than three vertices, then angle α may not be constant along whole length of a common edge.

8.1.1.6 Draw Circles

Radius of circle is equal to chamfering length, whereas its plane is given by a centre-point and a normal vector. Centre-point is located at ends of common edge and vector is identic with a tangent vector of common edge at centre-point.

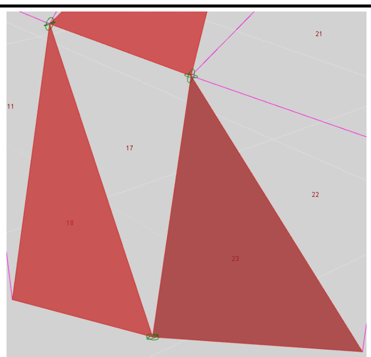


Figure 8-22: Draw circles Chamfering length = circle diameter. Plane of circle is given by a point (edge-ends) and tangent vector of the edge at this point.

8.1.1.7 Sweep Cylindrical Surfaces through Circles

Each pair of circles belonging to one edge is used as section-curves of a swept surface, while the edge itself is used as a rail.

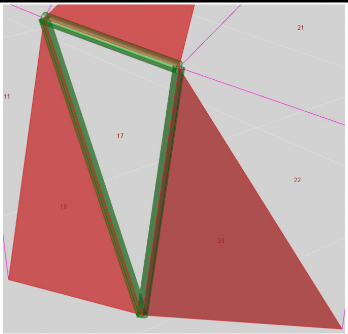


Figure 8-23: Sweep	Cylindrical surface is swept between two circles (one at each end) with edge serving as a rail (rail is important in cases where basic polygon has more than 3 vertices).

8.1.1.8 Intersect Cylinders with Surfaces

Title speaks for itself. Resulting curves are our sought-after keder-SLs, where distance between two neighbouring curves is equal to width of extrusion.

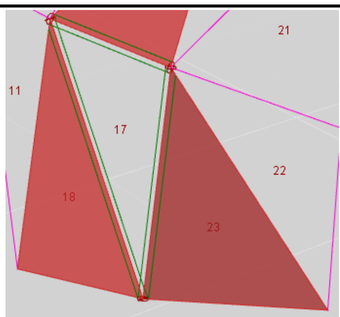


Figure 8-24: Intersect	Cylinders are intersected with both picked and neighbouring surfaces. Resulting curves are **keder SLs,** or **filling SLs.**

8.1.1.9 Restart with N+1

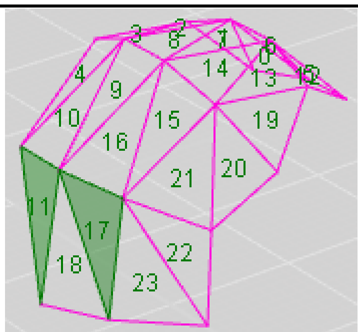


Figure 8-25: Restart with N+1	After completion of a last step, script starts again with $(N+1)^{th}$ surface – here the 18^{th}.

8.1.1.10 Problems with Sweeping Cylindrical Surface

For basic polygons with more than 3 vertices, some system-lines were significantly outside normal tolerance. Had inspected this problem, I can say, that it is a software problem of Grasshopper itself. That is, if two circles have different rotation with respect to rail (edge), then the cylindrical surface is twisted. Its shape could be compared with a twisted tin can (See Figure 8-26**Fehler! Verweisquelle konnte nicht gefunden werden.**). This doesn't happen, if the sweeping operation is done manually in Rhino. That is why it is an issue of Grasshopper. Currently, we are waiting for a response from Grasshopper support.

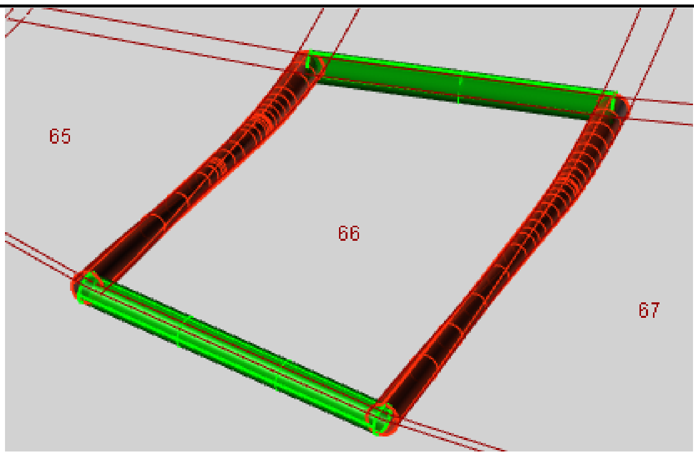


Figure 8-26: Twisted cylinders	Twisted cylindrical surfaces (red) and correctly swept cylindrical surfaces (green)

VI. Reprint of Parametric Script of Keder SLs Generation

In the Figure 8-27 you can see the final script. Red section generates cushion-SLs, violet section runs a loop by which it selects a surface by surface to be calculated and, finally, green section check precision of generated cushion-SLs and provides preview of those within and outside the tolerance.

Red section basically consists of five parts:

1) Input pole (top part, in the sense of this document)
2) Cluster "Pick'n'Find". It represents an algorithm which finds neighbouring surfaces to the one picked by violet section.
3) Cluster "Chamfer" which runs chamfering function with a set of surfaces provided by cluster "Pick'n'find".
4) Cluster "Filters" which filters duplicate cushion-SLs
5) Component "Filtered cushion SLs" (top left part, in the sense of this document), which previews what it says. At the same time, this is the input for green, checking, section.

In green section we can identify three parts:

1) Input pole (top part).
2) A group of clusters (dark blue) which checks distance between two neighbouring cushion-SLs and comparing it to chosen extrusion width with selected tolerance.
3) Output components (top left) "Curves OUTSIDE the tolerance" and "Curves WITHIN the tolerance" which preview what is says.

In following figures you can see those clusters in their exploded form.

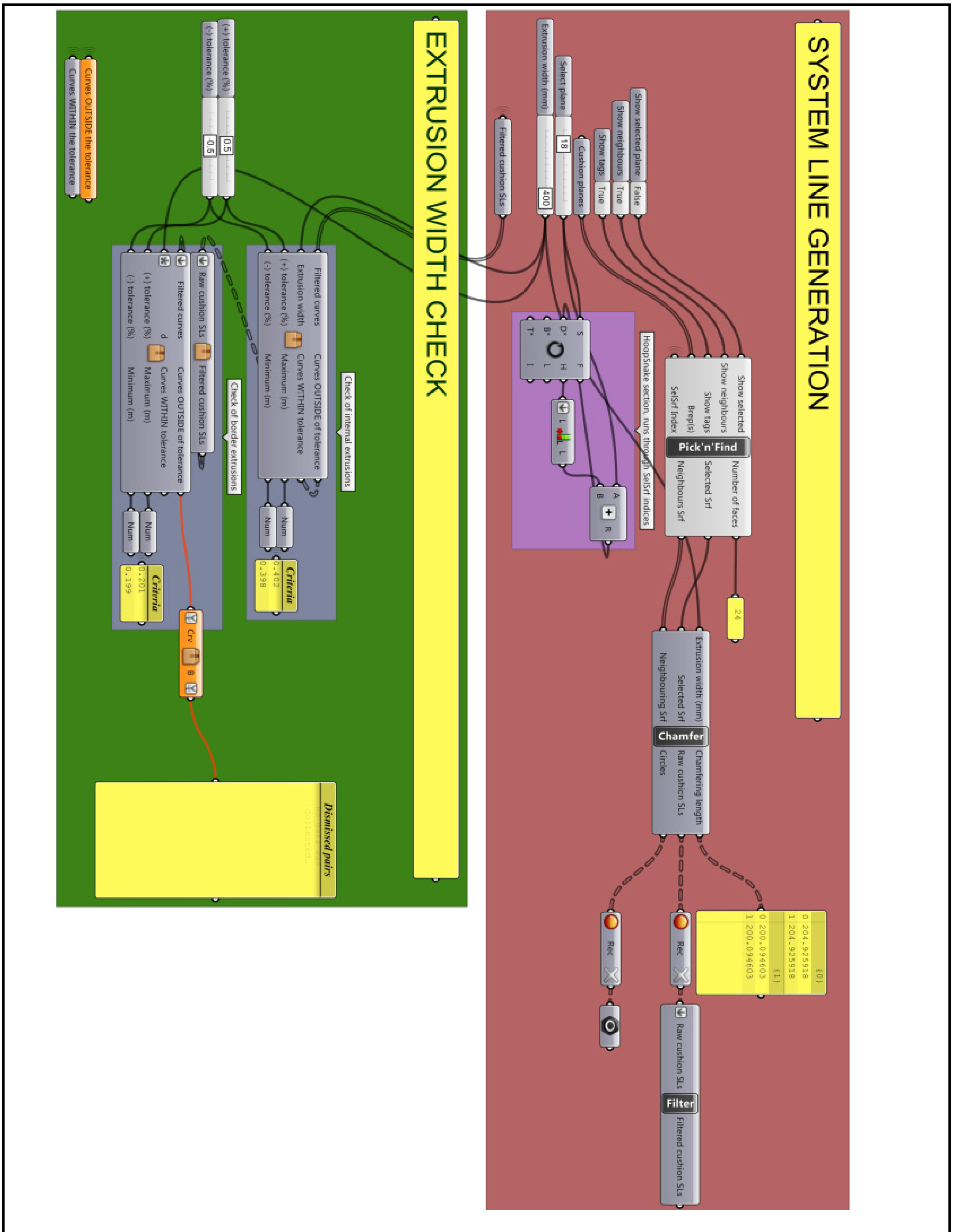


Figure 8-27: Overall View	Grasshopper Parametric Script. Red part generates keder-SLs, green part checks their precision by given tolerance (now set to +/- 0,5%).

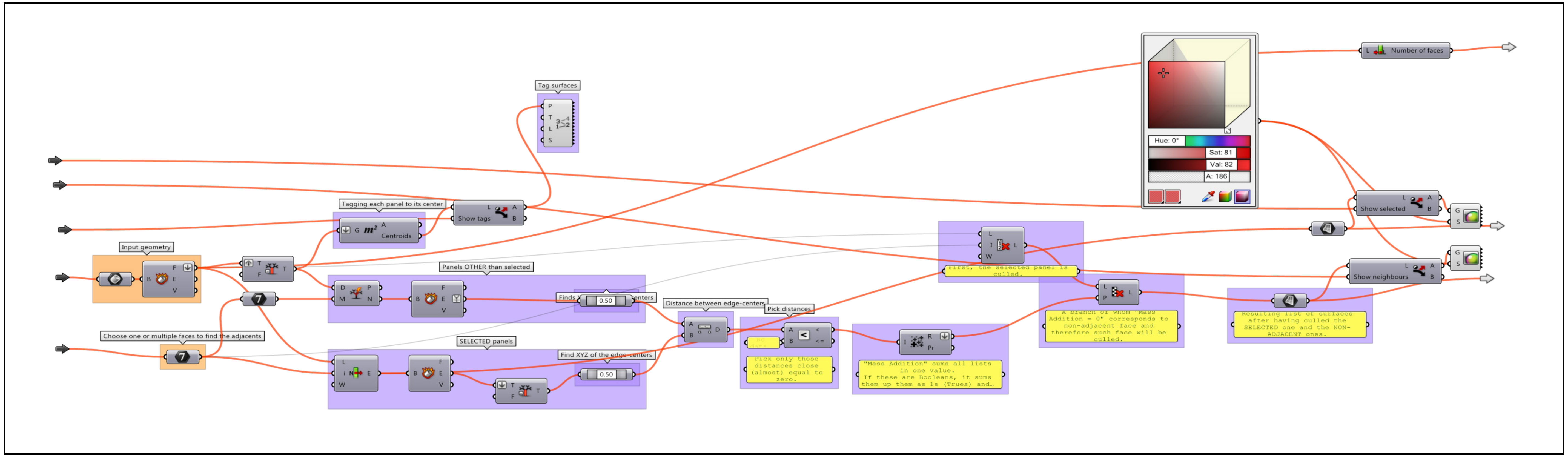


Figure 8-28: **Cluster "Pick'n'Find"**	Exploded cluster with tags and notes, so that it is easier for author to orient in the structure of it.

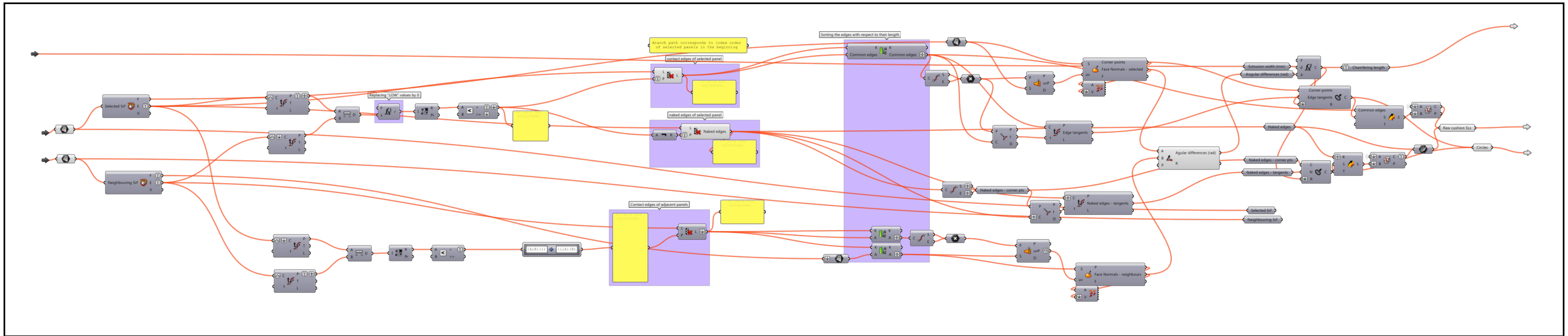


Figure 8-19: **Cluster "Chamfer"**	Exploded cluster with tags and notes, so that it is easier for author to orient in the structure of it.

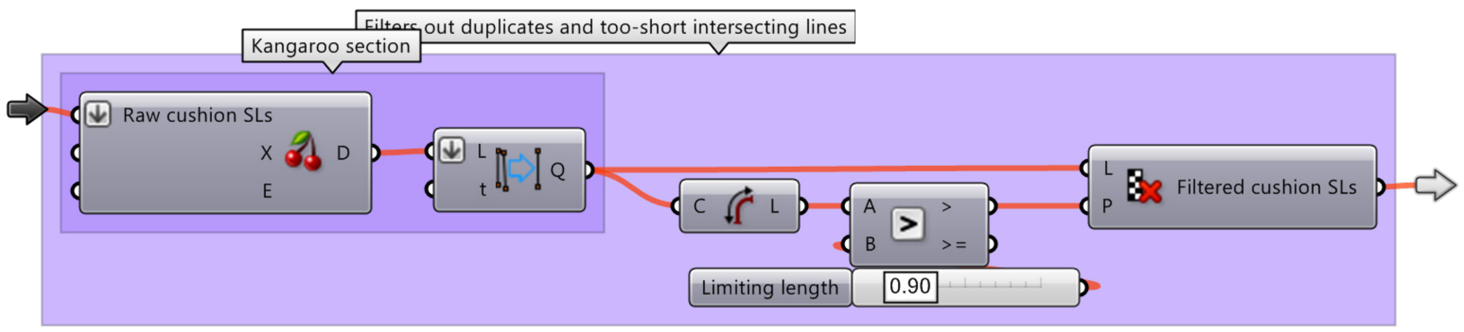


Figure 8-30: **Cluster "Filter"**	Exploded cluster with tags and notes, so that it is easier for author to orient in the structure of it.

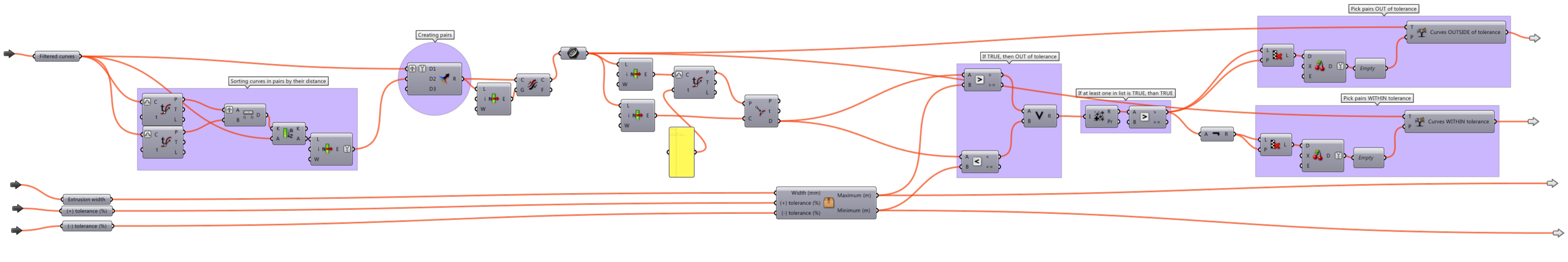


Figure 8-31: **Cluster "Tolerance"**	Exploded cluster with tags and notes, so that it is easier for author to orient in the structure of it.

VII. Reprint of Parametric Script of Data Preparation

In the only figure in this annex, you find reprint of a Grasshopper script mentioned in "5.2.2 Principles of the Parametric Script" on page 49.

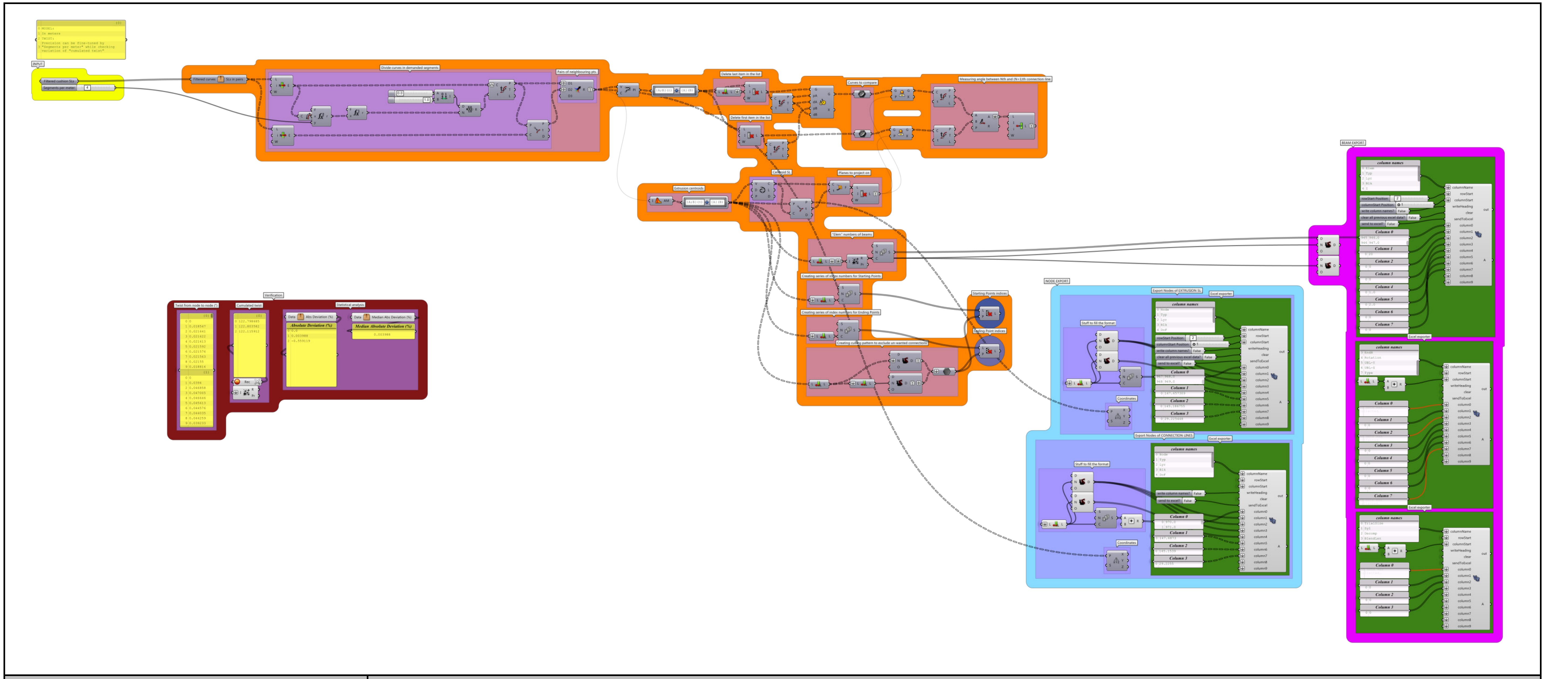


Figure 8-32: Parametric script for Chapter 5	Script that prepares geometric data, then it verifies precision and in finally it exports data to Excel spreadsheet.

Legend of coloured groups:

- Yellow – inputs: filtered keder SLs
- Orange – preparation of data
- Dark Red – verification of prepared data
- Cyan – export of node data
- Pink – export of beam data
- Dark Green – exporting units