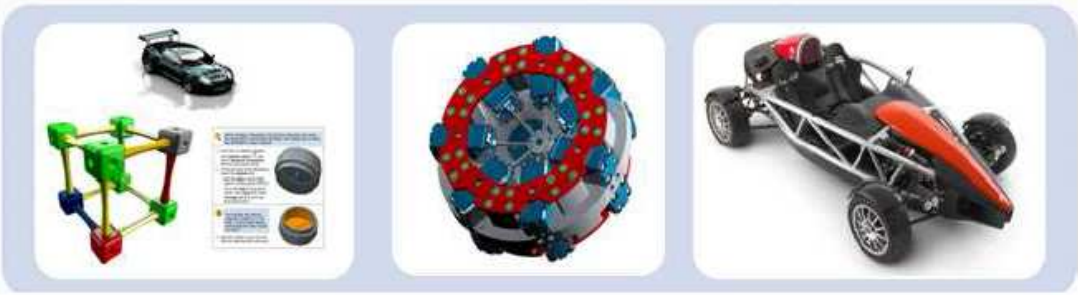
MÄLARDALEN UNIVERSITY
SWEDEN

IDT INNOVATION DESIGN AND ENGINEERING

KPP 307-Project Course

Product and Process Development



Autor: Gabriel Anaya García

Director de proyecto: Bengt Erik Gustafsson- *MachineTechnician; Division of School Administration*

Examinadores: Anders Wikström- *Lecturer; Division of Information, Design and Organization*

Magnus Wiktorsson- *Senior Lecturer*

Coordinador EINA: Francisco Javier Martínez Gómez

Índice

Parte I

Parte II

Adjunto: Tabla evaluación intermedia, CREO 2.0

Objetivo y alcance del proyecto

Objetivo

Introducción

-El presente proyecto se incluye como asignatura semestral dentro del bloque común de cursos de la Escuela de Innovación, Diseño e Ingeniería (IDT); órgano perteneciente a la Universidad de Mälardalen, con campus situados en las ciudades de Vasteras y Eskilstuna. La principal área de estudio del proyecto es el Desarrollo del Producto y del Proceso Productivo. Debido a ello, los alumnos integrantes del mismo pertenecen a titulaciones de Diseño Industrial, Ingeniería Mecánica y Desarrollo del Producto. Este proyecto no se incluye dentro de la oferta de cursos para estudiantes extranjeros de la Universidad de Malardalen.

Objetivo general

-De forma general, este tipo de proyectos se centran en la realización de un producto hasta su fase final, de forma que sea comercialmente viable. No obstante, ocasionalmente se centran en un estudio específico sobre algún tema que afecte directamente a la Universidad, de manera que el proyecto no es sino una investigación para obtener la información deseada y tomar decisiones en base a ella.

Contenidos y objetivos de aprendizaje

-Los contenidos que se espera que los alumnos desarrollen durante la realización del proyecto son los siguientes

-Aplicación las bases del proceso de desarrollo del producto en un proyecto real.

-Creación de grupos de trabajo y elaboración de documentación interna.

-Elaboración de diagramas de Gantt y planificación del proyecto.

-Documentación del proyecto, reuniones y trabajo en equipo.

-Uso de herramientas y métodos para el Desarrollo del Producto.

-Aportación de planos de componentes comerciales existentes, si procede

-Documentación oral y escrita del desarrollo de un producto comercial.

-Tras la finalización del proyecto, se exige que los alumnos pertenecientes al mismo sean capaces de:

-Presentar, argumentar y documentar de forma oral y escrita el proceso de realización y el resultado obtenido tras la realización de la tarea.

-Utilizar el diagrama de Gantt como base para la planificación de tareas y la duración de éstas durante antes y durante la realización del proyecto.

-Utilizar métodos y herramientas adecuadas para el desarrollo del producto.

-Utilizar herramientas CAD para la realización de modelado de 3D y planos necesarios.

Objetivo específico

El objetivo específico del presente proyecto es elaborar una estudio sobre las herramientas de software Creo 2.0® y Windchill 10.0® mediante la realización de un proyecto real, y así evaluar si son o no recomendables para su uso en la universidad, tanto a nivel de departamentos como a nivel docente.

Alcance

Utilizando como fuente de información principal los resultados obtenidos por el presente proyecto, la Escuela de Innovación Diseño e Ingeniería de la Universidad de Malardalen tomará la decisión de empezar (o no) a utilizar ambos programas para la realización de proyectos a nivel docente y a nivel de departamentos.

Introducción al tema

Fase previa

Durante los últimos años, los departamentos de Diseño del Producto e Ingeniería Mecánica de esta escuela han venido utilizando mayoritariamente el programa SolidWorks® para todas las tareas que requirieren herramientas de diseño CAD 3D. Este uso se hace extensivo tanto a nivel de proyectos internos, es decir, a trabajos realizados por profesorado o personas vinculadas a la facultad y enfocados a clientes externos o trabajos propios, como a nivel docente en los cursos y asignaturas impartidas en dicha Escuela. Cabe destacar que la Universidad probó en su día la plataforma ProEngineer®, como posible alternativa al uso de SolidWorks®. No

obstante, el personal universitario no quedó convencido con el rendimiento del programa, encontrándolo poco intuitivo e ineficaz.

Tras el lanzamiento de la nueva plataforma de CAD Creo 2.0®, evolución del programa ProEngineer®, la empresa editora del programa (PTC®) contactó con la Universidad de Malardalen con el objetivo de mostrar su producto. Según PTC®, el nuevo software había corregido todos los errores de su predecesor, incorporando nuevas herramientas de diseño y mejorando el manejo del programa. Además de esta plataforma, PTC® desarrolló paralelamente un programa de Product Lifecycle Management (PLM), denominado Windchill 10.0®.

Antes de introducir ambas herramientas, la Escuela decidió probarlas por medio de un proyecto llevado a cabo por un grupo de alumnos.

Personas involucradas en el proyecto

Personal Docente

Anders Hellstrom- Responsable de departamento

Bengt Gustaffson – Profesor Responsable del Proyecto

Grupo de trabajo

Fredik Larsson

Anton Lindberg

Nur Sadik

Daniel Lissert

Dina Thamir

Christian Eriksson

Bashar Bash Mansoura

Gabriel Anaya García – Project Manager

Los programas

Creo 2.0

Creo 2.0 es un programa de diseño CAD, desarrollado para llevar a cabo proyectos en 2D y 3D. Se divide en varios productos diferentes pero interrelacionados, cada uno con diferentes aplicaciones de diseño y desarrollo del producto. A continuación se enumeran los más importantes:

Creo Direct 2.0

Creo Parametric 2.0

Creo Layout 2.0

Creo Simulate 2.0

Creo Thermal

Creo Structure

De entre todos ellos el presente proyecto se va a centrar en Creo Parametric 2.0, programa de modelado sólido basado en características paramétricas. Es el producto más general y con mayor rango de utilización.

Windchill 10.0

Windchill es un programa de PLM, pero ¿qué es exactamente PLM?

Product Lyfecycle Management (PLM) se denomina al proceso de administración del ciclo de vida de un producto desde su concepción inicial, pasando por la fase de diseño, manufactura, comercialización, vida útil y retirada para reciclaje o eliminación. Este proceso se centraliza a través de herramientas software, y ofrece los siguientes beneficios:

- Reducidos tiempos a mercado
- Productos de mayor calidad
- Menores costos de prototipo
- Ahorros a través de la reutilización de datos originales
- Provee un marco para la optimización de productos
- Ahorros a través de la completa integración de flujos de Ingeniería

¿Cómo funciona Windchill 10.0?

Windchill posee un servidor que almacena los archivos pertenecientes al proyecto, ya sean partes, conjuntos, dibujos, BOM, etc. Cada archivo incorpora varias etiquetas que lo cataloga y describen la fase de desarrollo en la que se encuentra. Los miembros del proyecto tienen un grado de acceso que depende del administrador.

Cada persona integrante del grupo o empresa posee un espacio de trabajo independiente dentro del servidor, conectado directamente con el software que está utilizando (en nuestro caso Creo Parametric 2.0) y al que sólo ella tiene acceso. Una vez terminado el trabajo, el archivo se puede añadir a la carpeta común del servidor de forma que otros miembros puedan abrir y utilizar el archivo. No se puede modificar un archivo compartido si alguno de los miembros está trabajando en él.

Enfoque del proyecto

-En un principio, se nos presentan dos opciones para la realización de la evaluación:

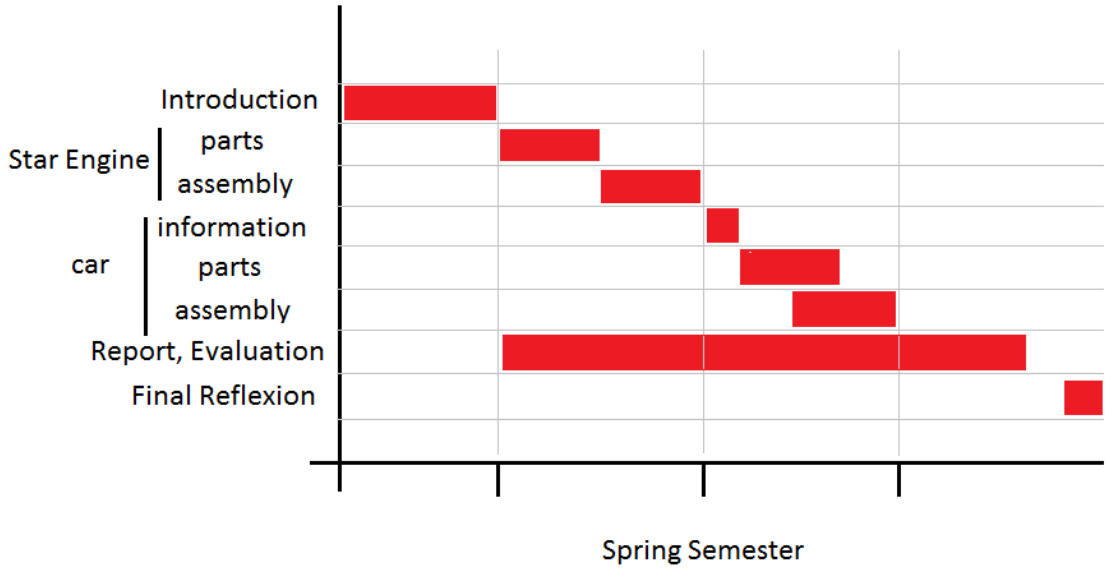
La primera de ellas es hacer un chequeo exhaustivo de cada herramienta del programa, y elaborar un informe que describa el proceso y los obstáculos encontrados. No obstante, este método tiene el inconveniente de no aportar perspectiva alguna. Es decir, al analizar las herramientas de forma individual es difícil que nos encontremos con los problemas que probablemente nos encontraríamos al realizar un proyecto, ya que en este último caso la utilización de comandos y herramientas de forma enlazada puede hacer aparecer dificultades que ni se nos ocurrirían en un análisis por separado.

La segunda opción es la de desarrollar un proyecto real. Esta opción ofrece varias ventajas: al realizar un producto desde el principio, vamos profundizando en el programa de forma gradual, lo cual facilita el análisis ya que al llegar a comandos de nivel avanzado ya tenemos una experiencia previa, que nos permite enfrentarnos a él más preparados; además disponemos de una buena oportunidad de aprendizaje, simulando el procedimiento de Diseño y Desarrollo del Producto que probablemente nos encontremos el día de mañana en empresas del sector industrial.

Por las razones citadas anteriormente, el método utilizado para realizar la evaluación de ambos programas es el desarrollo de un proyecto real.

Planificación del proyecto

-El proyecto se dividirá en varias fases a lo largo del semestre. Se utilizará el siguiente diagrama de Gantt como herramienta de control de plazos y fases del proyecto.



Estructura del proyecto

Fase inicial

-Toma de contacto con los programas. Durante esta fase realizaremos tutoriales provenientes de la propia empresa editora de los programas. También asistiremos a breves cursos impartidos por personal externo a la universidad. El trabajo durante este período será individual. El objetivo de esta fase es darnos la soltura necesaria para continuar el proyecto por nuestra cuenta.

Proyecto Individual

-Realización de un producto de dificultad intermedia. También de forma individual, todos los miembros del grupo desarrollarán un objeto mecánico compuesto por varias piezas, pasando por los diferentes estadios de diseño: planificación y esbozo, diseño de las piezas y ensamblaje. Se utilizará Windchill como plataforma de almacenamiento. Al finalizar este proyecto el grupo procederá a la primera evaluación de los programas.

Proyecto Final

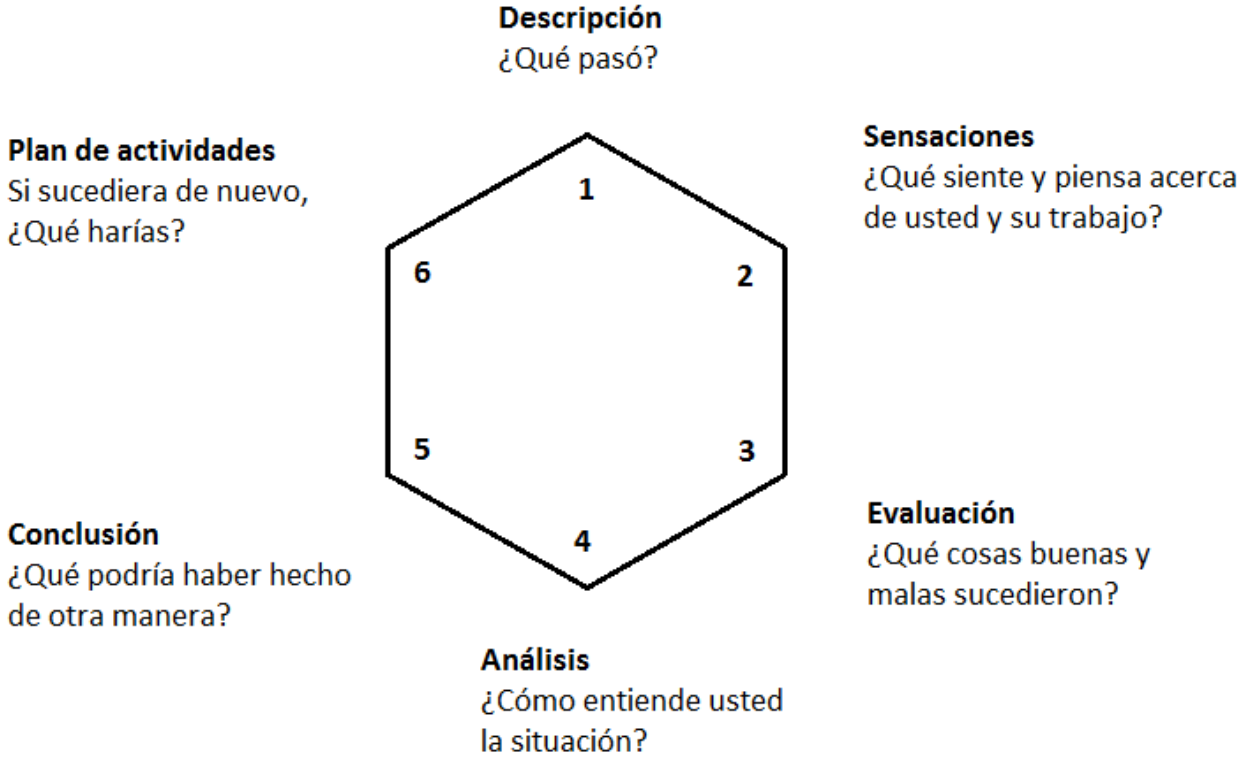
-Llegados a este punto, es de esperar que el trabajo previo haya otorgado a los miembros del grupo la capacidad necesaria como para emprender un proyecto de mayor dificultad. Durante esta fase se desarrollará un producto de forma conjunta, de manera que la planificación, comunicación y el intercambio de archivos serán indispensables. Esta será la prueba más importante para la evaluación de ambos programas, especialmente para Windchill. El informe acerca de este proyecto se adjunta en inglés.

Evaluación

-Finalmente, el grupo realizará la evaluación final de ambos programas. Se compararán los resultados con los obtenidos en la evaluación intermedia para apreciar si ha habido cambios, y por último vendrán las recomendaciones finales sobre ambos programas.

Reflexión final

-Algunos días después de finalizar el proyecto, cada miembro del grupo escribirá una reflexión individual acerca del mismo. En ella se reflejará la opinión global sobre el proyecto. Se realizará en base al ciclo reflexivo de Gibbs.



Desarrollo del tema

Fase inicial

Curso introducción a Creo Parametric 2.0

Impartido por: Marie Eriksson, estudiante de Máster proveniente de Universidad de Estocolmo.

Lugar: Escuela de Innovación, Diseño e Ingeniería. Campus Eskilstuna.

Fecha: 11/02/2013

Duración: 4 horas

Descripción: Breve introducción al uso de Creo 2.0. Explicación del manejo de sesión y archivos, visualización e interfaz del programa. Además toma de contacto con herramientas básicas de referencia como **Esbozo**, **Plano**, **Punto** y **Eje**; así como con herramientas de ingeniería como **Extrusión**, **Revolución**, etc.

Tutorial acerca de Windchill 10.0

Impartido por: personal proveniente de empresa de servicios informáticos contratado por la Universidad.

Lugar: Escuela de Innovación, Diseño e Ingeniería. Campus Eskilstuna.

Fecha: 18/02/2013

Duración: 8 horas.

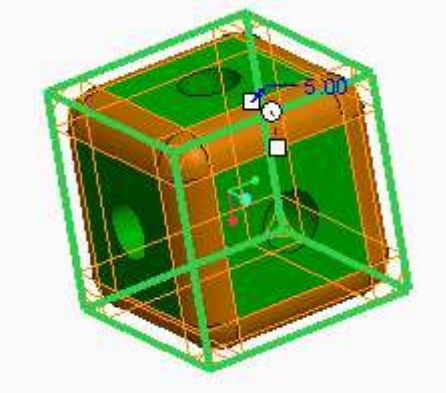
Descripción: Curso intensivo acerca del manejo de Windchill 10.0. Descripción de PLM, utilidad y aplicaciones. Crear servidor, crear cuenta de usuario, navegar por la interfaz, manejo de archivos, conexión con Creo Parametric 2.0.

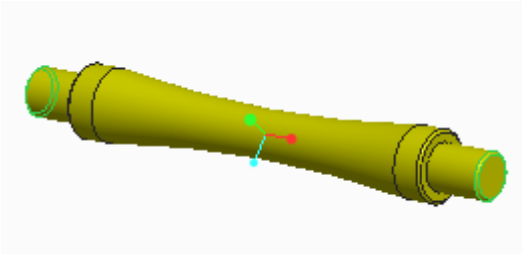
Creo Parametric Primer

-Construcción de un juguete con cubos y barras. Este tutorial se divide en tres fases durante las cuales se utilizan diversas herramientas. Está diseñado para usuarios principiantes que quieran adquirir soltura con los comandos básicos y empezar a comprender el funcionamiento del programa. A continuación se describen las fases del mismo: modelado de partes, conjunto y render.

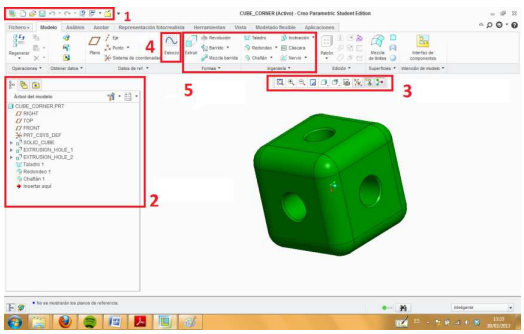
Modelado de partes

Para el modelado de uno de los cubos de los vértices y las barras intermedias utilizamos comandos como **Esbozo, Extrusión, Revolución,Redondeo, Chaflán y Taladro.**





Para el desarrollo de ambas partes utilizamos las siguientes herramientas:



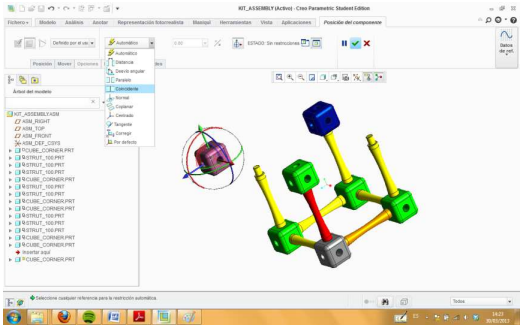
1-Opciones de Archivo (Nuevo, abrir, guardar,etc.).

2-Árbol del modelo

3-Opciones de Visualización

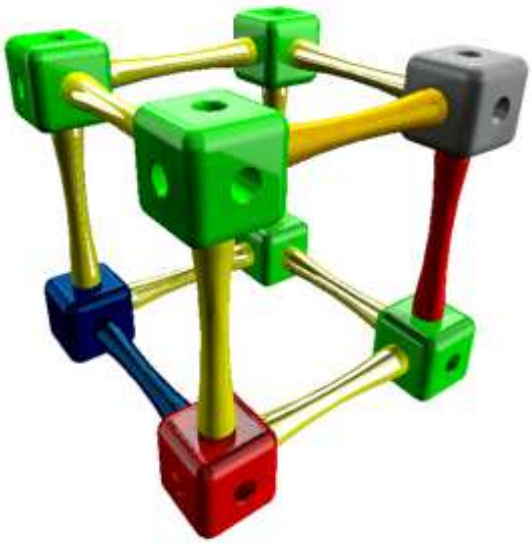
4-Esbozo

5-Aplicaciones de Ingeniería (Extrusión, Taladro, Revolución,



Conjunto

Durante la segunda parte del tutorial se introducen los comando y opciones para realizar conjuntos a partir de piezas. Cabe destacar que los archivos de parte y conjunto tienen diferente configuración.

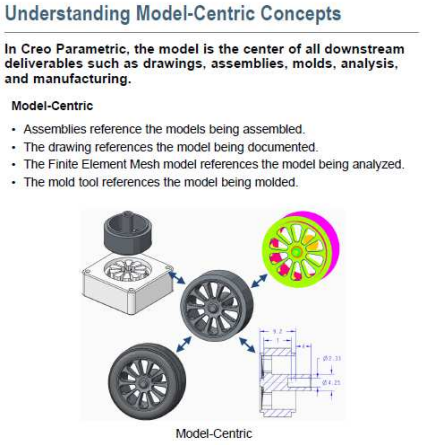


Render

La herramienta Render nos permite realizar visualizaciones realistas de nuestro proyecto, dándole un aspecto más atractivo mediante el uso de perspectivas, sombras, reflejos, etc. Es una herramienta muy potente, que en el caso de tener el ordenador y la habilidad adecuado hace posible imágenes de realismo casi fotográfico.

Creo Parametric 2.0 Advanced Premier *–Guía*

-Este tutorial muestra las posee un grado mayor de dificultad que el anterior. Está dividido en dos partes, la primera es una descripción general de las características del programa, proceso de instalación y configuración. El segundo explica como modelizar una parte, empezando por las herramientas básicas hasta otras de un nivel más avanzado.


Understanding Model-Centric Concepts

In Creo Parametric, the model is the center of all downstream deliverables such as drawings, assemblies, molds, analysis, and manufacturing.

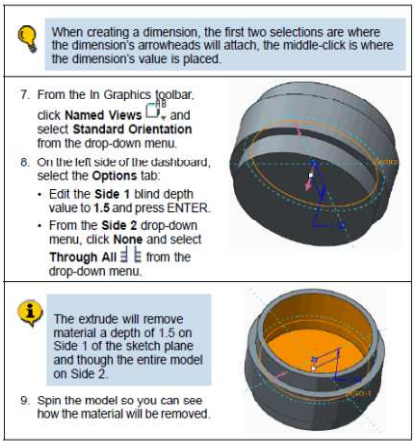
Model-Centric

- Assemblies reference the models being assembled.
- The drawing references the model being documented.
- The Finite Element Mesh model references the model being analyzed.
- The mold tool references the model being molded.

Model-Centric

First part: The interface and basic concepts

Este


When creating a dimension, the first two selections are where the dimension's arrowheads will attach, the middle-click is where the dimension's value is placed.

7. From the In Graphics toolbar, click **Named Views** and select **Standard Orientation** from the drop-down menu.
8. On the left side of the dashboard, select the **Options** tab:
   - Edit the **Side 1** blind depth value to **1.5** and press ENTER.
   - From the **Side 2** drop-down menu, click **None** and select **Through All** from the drop-down menu.

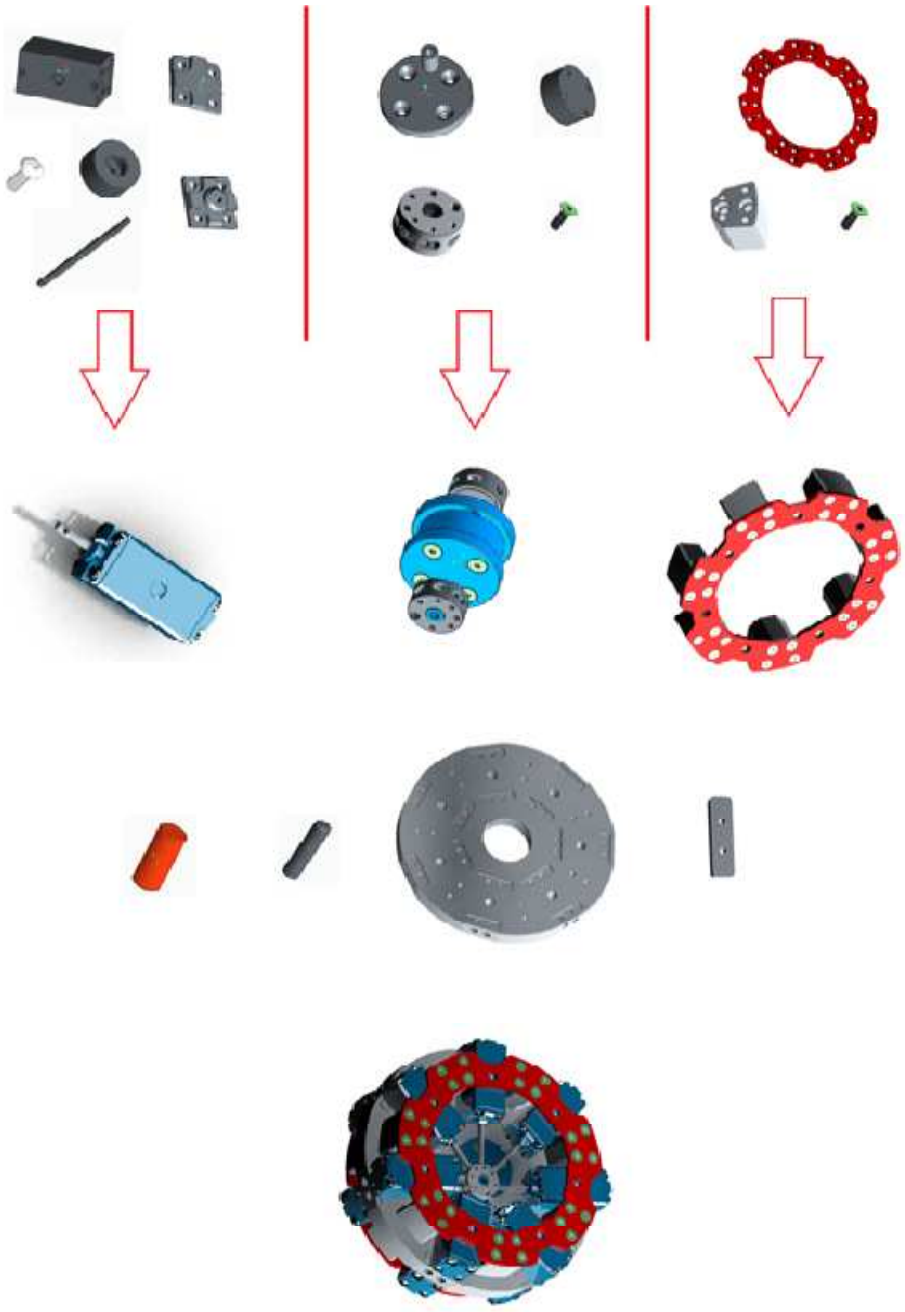
The extrude will remove material a depth of 1.5 on Side 1 of the sketch plane and though the entire model on Side 2.

9. Spin the model so you can see how the material will be removed.

Second part: drawing, part, assembly, rendering

Proyecto individual *–Motor Radial*

-Diseño de un motor radial a partir de planos de Solidworks. Realización de las piezas por separado y posterior ensamblaje de las mismas en subconjuntos y en conjunto final.



MÄLARDALENS HÖGSKOLA
ESKILSTUNA VÄSTERÅS

**Akademin för Innovation,
Design och Teknik**

KPP-307

Project Course, Product and Process Development

7,5 högskolepoäng, avancerad nivå

Gabriel Anaya

Presentationsdatum: 12 jun 2013
Uppdragsgivare: Mälardalen Högskola

Handledare(högskola): Bengt Gustafsson

Examinator: Anders Wïkstrom, Magnus Wiktorsson

Table of content

1. Introduction

The purpose of the courses KPP307, KPP309 and KPP318 is to learn the program Creo with a PDM system Windchill and then evaluate those. Eight students worked on this project together with the teacher BengtGustavsson.

To learn the programs the group got an introduction about the program Creo by a student who has previously worked with Creo and another introduction about the program Windchill by XX. This project will be about construction of a crosskart in the CAD-program Creo using the PDM system, Windchill. Eight students have collaborated and the "car" was split so that everyone had a part to construct.
This report will help the teacher to be able to evaluate the programs and contemplate if they will be used and taught in MDH. (underlag)

Uppdelningenär:

Anton: back suspension

Bashar: Interior and seats

Christian: Front suspension

Daniel: Engine

Dina: Wheels

Gabriel: Main frame

Fredrik: Main frame

Markus: Breaks

Nur: steering

2. Aims and objectives

The purpose of this project is that all course members will learn how to use the programs Creo and Windchill and to present the work for the teachers then write a reflection of how the programs worked. This was done by all course-members got a part of a crosskart to construct. The goal is, with the help of programs Creo and Windchill, and in cooperation with the course members, constructs a Crosskart.
After completing the course, the course members will evaluate the programs for Bengt Gustafsson.

3. Project directive

The directives for this project came because Mälardalens University wanted to evaluate whether to use PTCCreo as a compliment to Solidworks in later courses or not.

A comparison between Creo and Solidworks, and how they both can work together with Windchill, has been the goal for the project. The project should result in a presentation and a report telling the client (MDH) if Creo is believed to be a good compliment or not.

To be able to evaluate the programs a CAD-project should be set up where the students make different parts in Creo and share them using Windchill.

To do this a course was set up at 25% study time. This equals up to ten hours a week. A 25% time course runs over one semester and gives a participating student 7.5hp. As a starter the students should set up one weekly meeting to discuss and help each other.

4. Problem

Evaluate the 3D mechanical CAD (computer-aided design) program Creo 2.0 together with the compatible PLM-program Windchill. The evaluation is desired to reflect over the possible benefits for MälardalensHögskola if purchasing these products and using them in everyday education.

The evaluation shall act as a good basis for future decision making regarding the purchase of the aforementioned products.

Fields of interests are particularly:
> ComparisonSolidworks versus Creo 2.0. With focus in features used by post-second year and senior students.
>Handling CAD-files within in Creo by using the Windchill platform. With focus in if better ways of information sharing and easier managementin complex product development chains are achieved.

> Compatibility between Solidworks and Windchill.

5. Boundaries

This course will be conducted during the 20-week period corresponding to 7.5 credits. Group aims of this work are to learn to use CAD programs Winchill and Creo.

The project will result in an assembly of a crosskart which will be developed by the group members. Each and every one group member has constructed a part of the product. a report will be written by the group members to clarify and document the group's work. The group has decided to meet three times a week, totally 10 hours per person.

The Crosskart will be constructed with Creo and each group member has the responsibility to find information for his or her part and construct it in Creo, during a specific timeframe. The group will assemble them together at the end of the course.

A limitation on the choice of materials will be made since the main objective of the project is to learn how to manage the program. Each team member will be responsible for his or her part both in cad Creo and in the report.

6. Process

Mention shortly that we were a group of 10 people, we decided to do an Ariel Atom because it seemed reasonably since it requires many functions in order to design it etc… We divided thecar into X-amount of parts and each group member was responsible for X-amount of parts and designed them.

6.1 Steering wheel – *Bashar Mansour*

For the Ariel Atom we needed a steering wheel with more complexity in comparison to the original steering wheel for the Ariel atom. Why more complex? Since this project was about evaluating the program Creo Parametric, models must be complex in order to try out the different features including the software. Below is a picture of the original steering wheel for the Ariel Atom and to the right side is the result from this project.

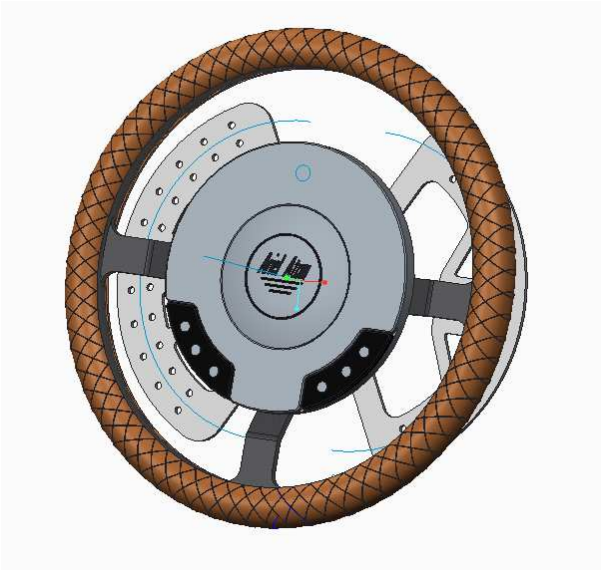
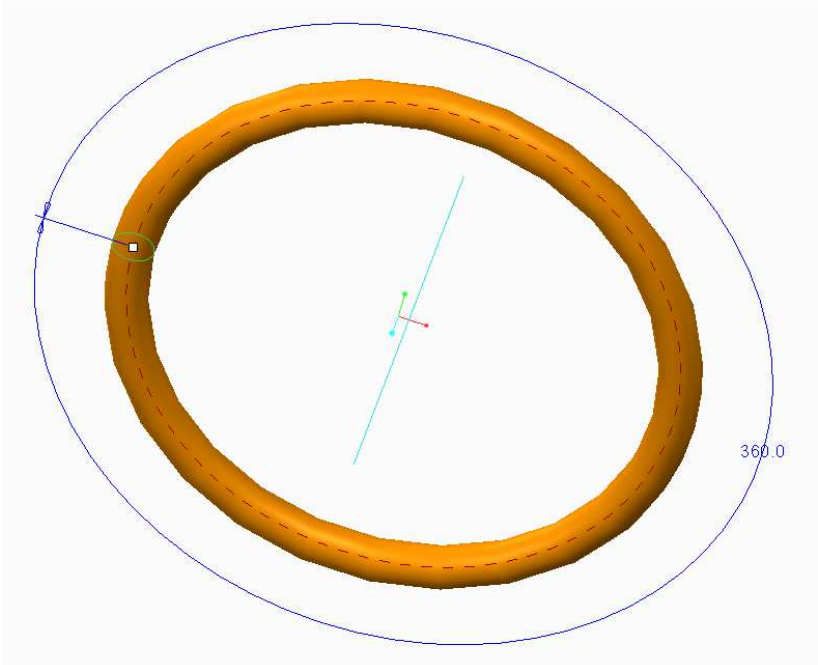


Figure 1: Steering wheel, Ariel Atom

The original Ariel Atom uses manual gear box like normal cars with normal shift. For this project the group decided to use shifting paddles mounted behind the steering wheel. With the shift paddles, it looks better and adds more details to the steering wheel. The aim with this model is to use enough features in Creo so that a good and fair evaluation can be done, there for it was not important to design for manufacturability.

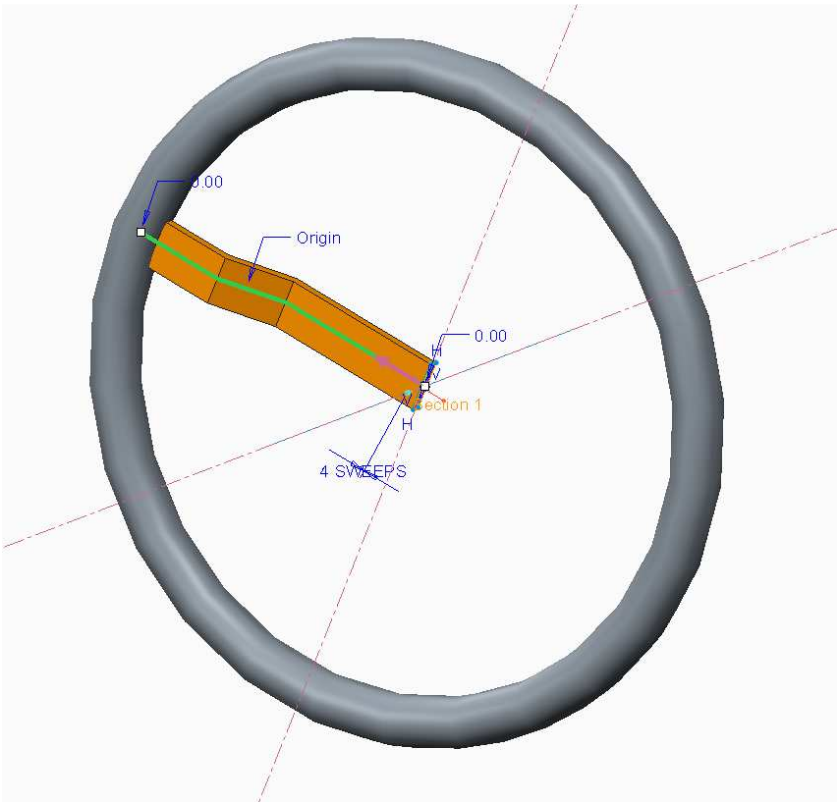
Tools used

- **Revolve**
  This tool was used to make the wheel. A circle was sketched with distance to an axis to revolve around. Measures were taken from internet for standard steering wheels and half of that diameter measure became the distance from the circle sketch to the axis. Using the Revolve feature was easy, you only need to select sketch and select axis and then it revolves. It is easy to select witch angle to revolve, 90, 180, 270 and 360 degrees are in standard to choose from but you can type in exact angle manually. Good tool and very easy to use.

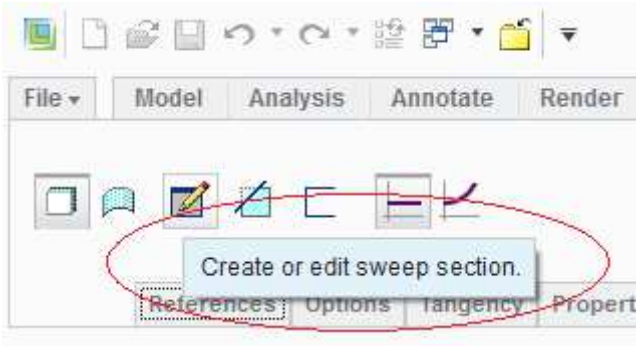


- **Sweep**

This tool was used to create the first body of the three spokes. See picture below.



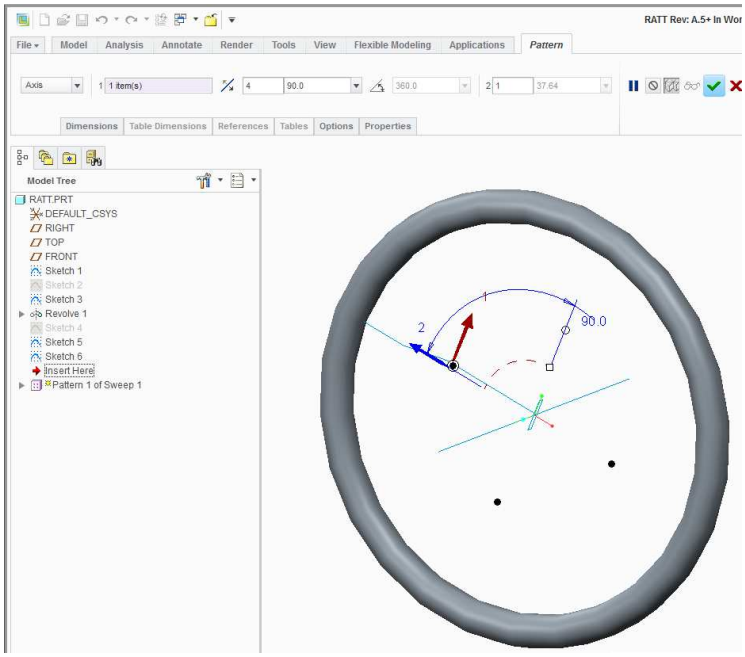
At first the user must sketch a “track” for the profile. After sketching the track you click on the sweep icon and then you get the options to select profile to follow the track. In my experience it was difficult. Sweep did only succeed if you choose to sketch profile while you are in the sweep tool. It always happened that tracks and profile were sketched, but profiles sketched once again in sweep tool. It’s possible to use pre-sketched profiles but it´s not easy to find. In comparison to Solid Works CAD-software, the user only need to click on the sweep tool, select track and profile and press ok. Below is a picture showing pop-up message for creating or “edit” sweep section.



As mentioned above, this feature only works if the user chooses to create “sweep section”, which means profile. Good tool but not easy to use and everything feels to be hidden from the user. It is not user friendly.

- **Pattern**

For the rest of the spokes to the wheel, the pattern tool where used. Like the steering wheel for the Ariel Atom, this steering wheel was also about to get three spokes. In order to use pattern tool, you need to select the features you want to pattern and then press on the pattern tool. After selecting the first sweep that was made to do the first spoke, pattern tool was used and the options you get looks easy to use. You can select to pattern around an axis etc. you can insert how many copies you like to have and what angle to distance between them. Since this steering wheel only needed three spokes, and the first one was made, it only needed to pattern to copies. This tool made me insert 4 copies. It was crazy to find how to remove selections. The black dots are actually “checkboxes”. You can click on a black dot to remove that “instance”. See picture below.



This tool is not easy to handle and if you want to add more features to the same pattern you need to cancel the “pattern tool” and start over again and mark all the features you want to pattern in same time, not good.

- **Extrude/cut**

This tool is probably the most common to use. The beauty in this tool is that you can choose to extrude sketches or use the sketches to cut. It is logic sins in different CAD software´s, you must choose a particular tool for cut and particular tool for extrude.

This tool was used for making the middle section where the logo and extra buttons is attached.

6.2 Seats – *Bashar Mansour*

The group decided to make this Ariel Atom with two seats and not one. The seat from the original Ariel Atom come in one-piece and is very advanced in its organic shape. However it must be done and does not have to look exactly like the original seats.

At first it looked like a difficult task, but one tool became the rescuer. The tool loft was used to accomplish one-pieced seat. The loft feature in Creo Parametric is called Swept blend.

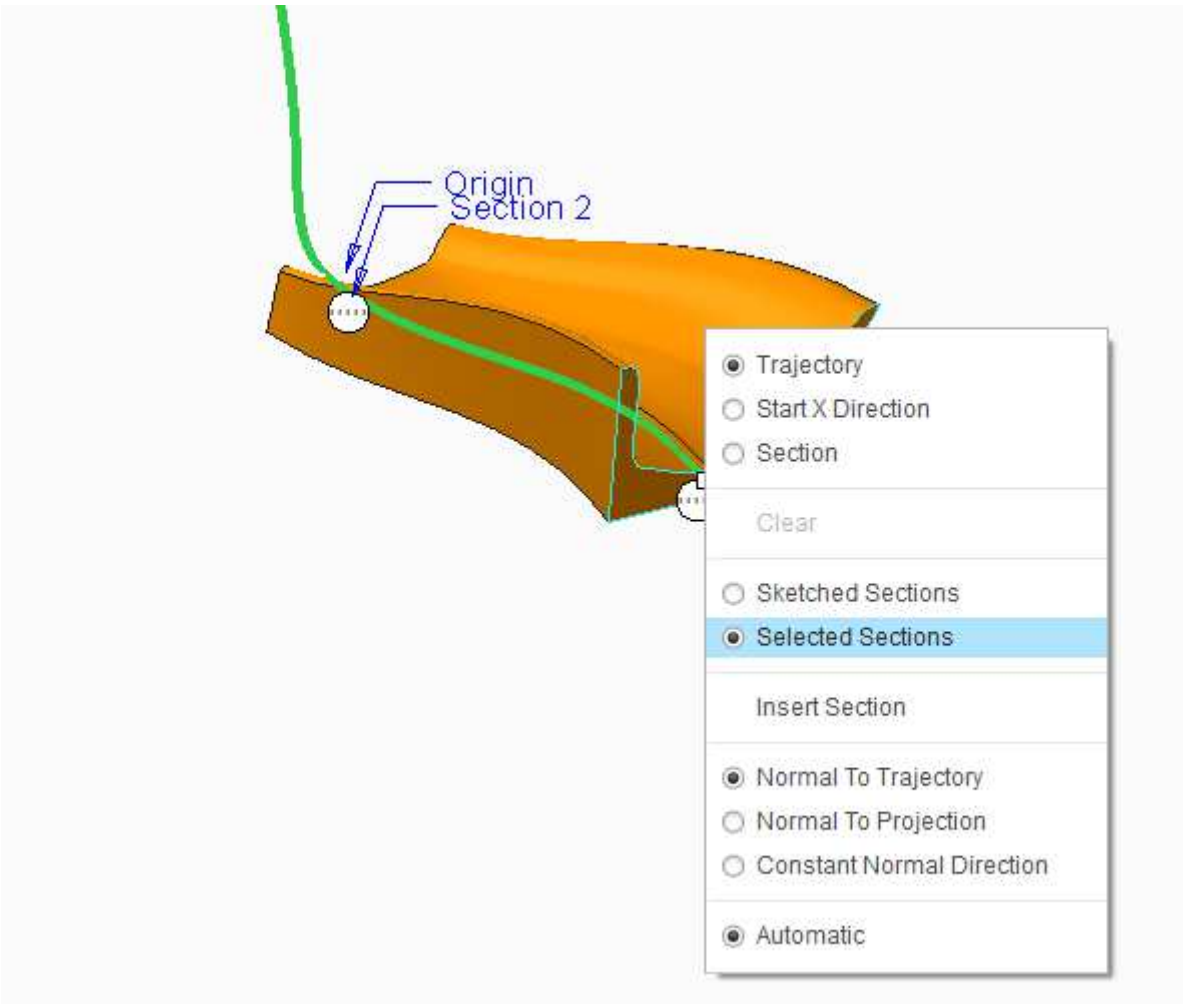
Swept blend is basically the tool to use when you want to model something that changes shape. For example seats. Car seats has bottom, back support, Leg-support and they cover parts of your shoulder. Car seat is a typical part that is changing shapes from one point to another. See pictures below to see the result.

To make this seat, seven sketches were used including one track. How does the swept blend work?

In order to start you need to sketch a track. See picture below (green line).

This track must be selected before pressing on the swept blend tool. After that comes the option to create or select profiles. The first-time-users will go mad about this tool. Unlike different CAD-software´s you can´t select a track and then just select profiles to follow the track and transform.

- All the sketched profiles must have exact same amount of breaking points, or it will fail totally.
- You must first have selected at least one sketch in order to be able to use swept blend.
- You must create all the sketches for the profiles before entering the swept blend tool.
- You cannot select and connect profiles directly; you must right click and select “Selected section”.



- In order to continue the swept, you must right click again and “insert section” etc. until you are satisfied.

If you master this tool, then it´s very effective but for normal users it’s more to do and the process is longer than necessary in comparison to Solid Works for example.

6.3 Steering –*Nur Sadik*

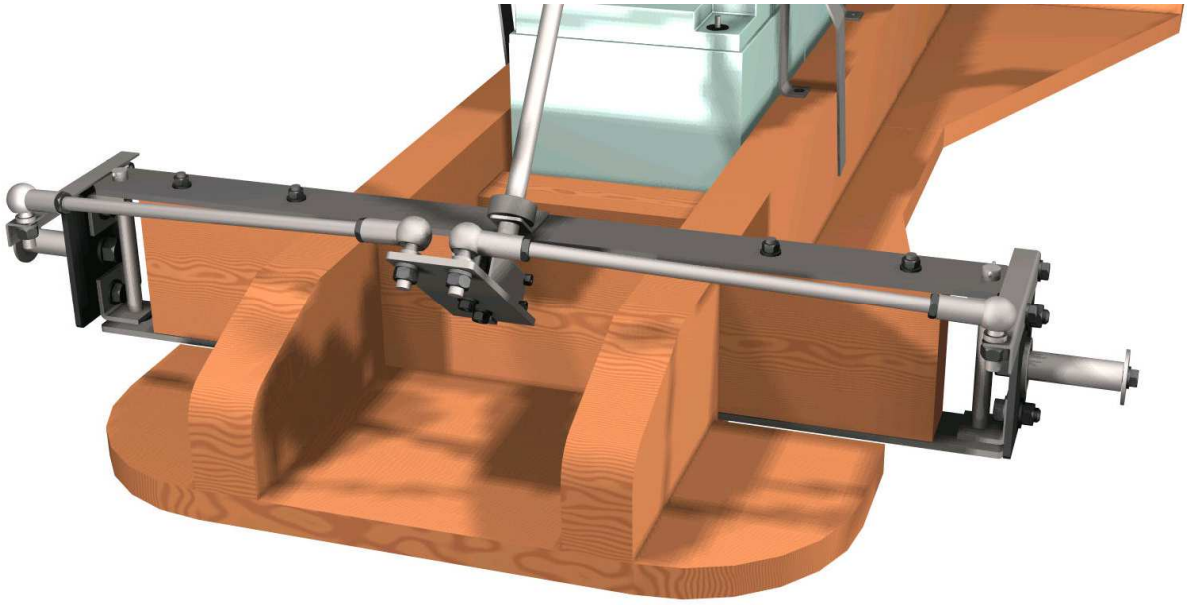
The group needed to design a control which and it will be located between the wheel and tire. I started searching information about similar models to get insperation.. I found the model below and decided to use it as a reference for my construction.

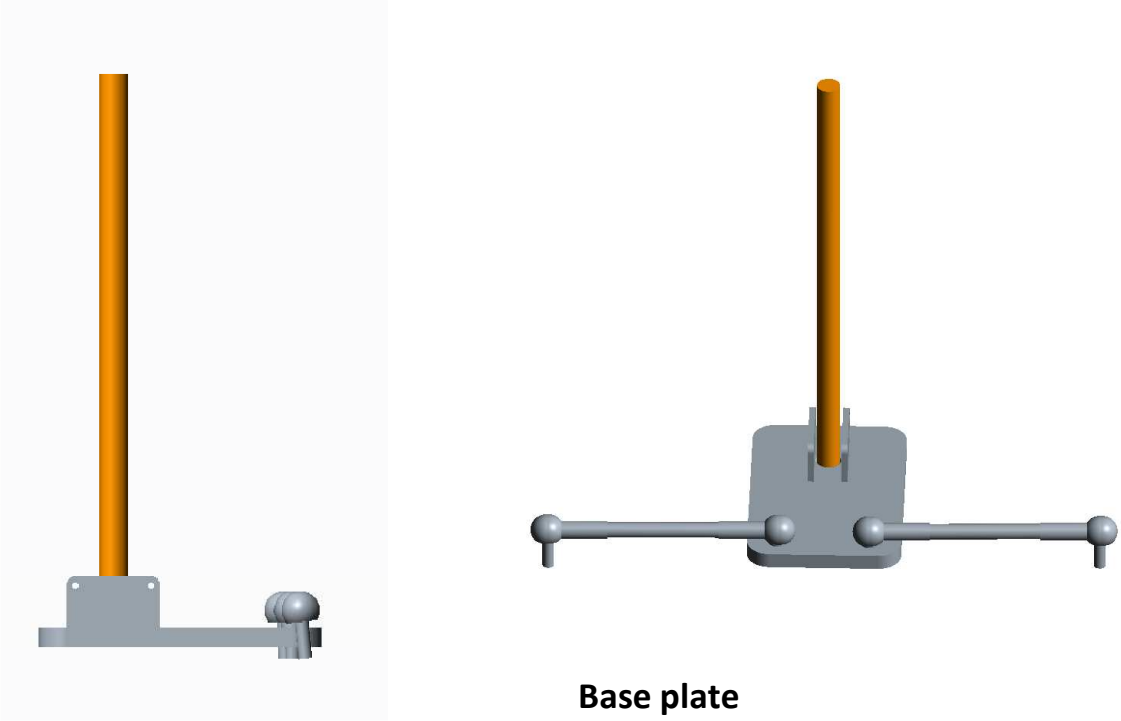
I constructed a similar model on Creo. The results of this part are shown below:

Overall picture

I started to constuct the mode lusing different tools, the result is three pipes, four locks and a base plate.

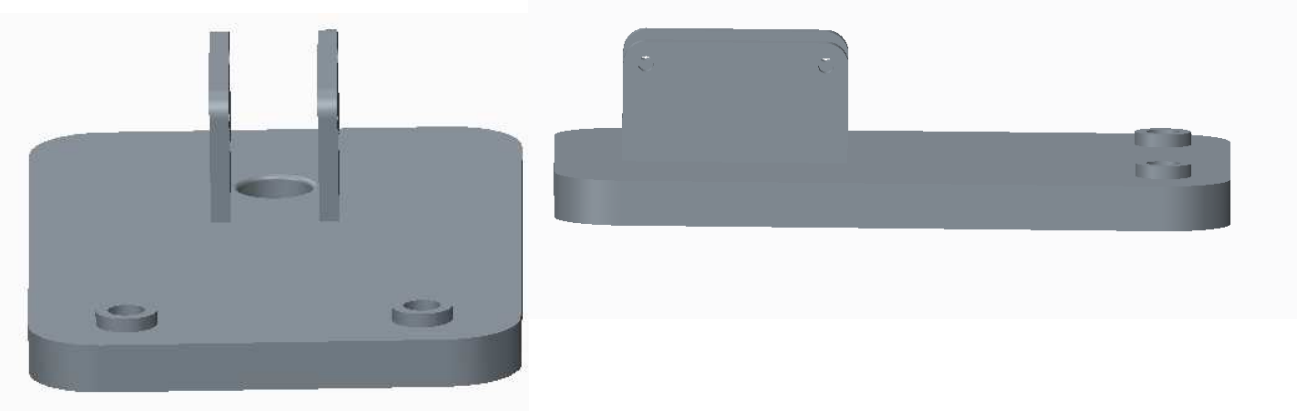
On the front of the base plate there is two locks on each edge. On the lock you screws a pipe. The lock is then mounted to the base plate and then to the front suspention.





Base plate

The base plate has a length of 300 mm, width 200 mm and depth 30 mm. There are three holes on the base plate to mount the other components.

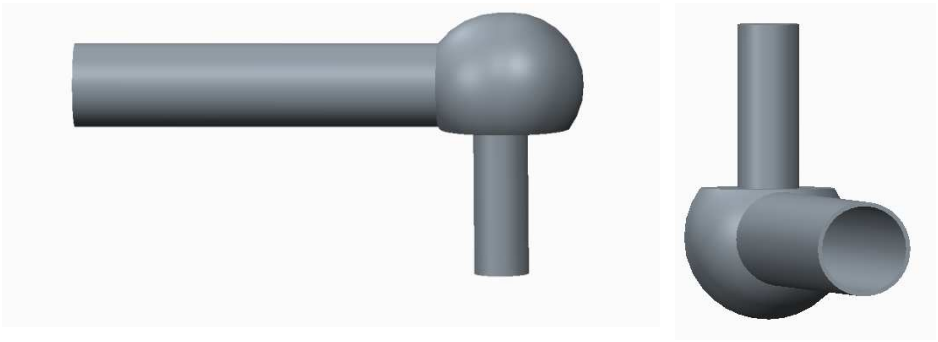


Stir

This pipe will be mounted on the steering wheel and the base plate. it has a diameter of 30 mm and length 600 mm. Then two pipes of 30 mm diameter and the length of each tube 300 mm mounted on the locks.

Lock

Four locks in the same dimentions are used to mount the other components to the base plate and the front suspention.



General Creo Sketch-tools

The tools used for the construction of the steering where: extrude, hole fillet and plane.

Advantages:

- • when i sketched circles on a the plane, it was easy to have the same dimentions of the circles in the same sketch.
- • easy to choose plan

Cons:

- • it is difficult to definne measure of all sketches

Extrude

This tool was used to make the tube to the control. A circle with a diameter of 30 mm was sketched and then using the extrude tool, the length of the pipe to 600 mm. This tool was easy to use and measure.

Extrude / Cut

I used this tool to make holes in the lock. Thie holes were made for installing the lock to the pipes.

When I constructed the various parts of the steering I used different functions and they worked well. But some tools were difficult to use because it was difficult to handle these tools.

Reflections on the tools

It was quiet easy to use extrude-cut to make the holes and create locks. I used the front plane for every part because it would not be a problem to put the parts together in assembly.

When using the fillet tool it was hard to dimention the part and it was difficult to do the mates in the assambley.

Assembly

The most difficult thing was at first when I tried to move the model. Because the parts were stuck and they did not move. But with then I got help of team members, we realized that it is possible to move parts of the asembly and suddenly it became much easier.

It was very difficult to assemble the parts to each other compare with SolidWorks. But the I had a basic part so it was quiet easy to assamble the part for the steering.

Conclusion

Because I had no knowledge of the steering of a crosskart it became difficult to use different tools and draw a complicated model of this part. It was also difficult to find a detailed model of steering model and that is why I chose the construction of the seering shown above.

I found it difficult to render images on Creo and select materials of the different parts of the model.

In the beginning of the course, I got to design an engine with 18 components. This helped me to understand how the various tools used in the creo. But when the group decided to make a model of crosskart and because of the simple construction of the steering I did not get the opportunity to use many different tools.

6.4 Engine – *Daniel Lissert*

When creating the engine a tutorial for Solidworks was used. The tutorial gave me the opportunity to directly compare the programs with one another. By using the same features in Creo as they did in the tutorial I had a step-by-step guidance throughout the design.

When creating the engine I used lots of basic features. These features worked fine, some even better (*read easier*) than in Solidworks. Although when it came to the more complex features my fait faded. It might have to do with that I have basic knowledge in another program which effects my conclusions, but the program did not meet the expectations.

Main tools used in the making of the engine:

Extrude, Hole, Round/ Chamfer, Mirror, Pattern, Group, Shell, Plane

My own reflections

The first problem I did get was that I was unable to use the Extrude-feature, extrude up to surface, then there was a plane between my sketch and my "target surface". Terrible in all ways, the program must offer this feature.

Holes were hard to put in place and the different types of holes where hard to understand. The standard holes had a small diameter and to "activate" larger ones you had to change the standard settings, and that it's easy if you know it.

When using the Round-feature some problems emerged. Sometimes you couldn't choose more than 4 to 5 edges in the same run. And if you wanted to change the radius on an already set round you would need to change all the measurements separately, truly ineffective.

(Same goes for chamfer)

At one time I used the Mirror-feature. The chosen part was a bit complicated but nothing like a "real-work-part". The feature created the mirror but to my surprise I was not able to extrude any geometry up to its surface, the new piece acted like a "shadow-copy" of the original. I'm not sure that I got the intended result it could have been a bug in the program, but if that is how the feature works it's a disaster.

Another bad thing about the worth mentioning is that the Mirror-feature don't give you a preview of the result, which is desirable.

The Pattern-feature worked fine. I was able to do what I wanted and I had the basic settings needed to control the results, a good feature.

General about Creosketch-tools:

+ When sketching multiple circles you can easily set the same diameter for all the circles.

+ When sketching a circle you'll get access to the measure when releasing the sketch-tool.

+ When using a sketch-tool you can release the current sketch and start over with the same tool, effective when you don't need to reselect if you started at the wrong place.

- Hard to know if all measurements needed are set. (fully defined sketch)

- When exiting a sketch measurements is automatically set and often creates undesirable relations.

6.5 Exhaust pipe – *Daniel Lissert*

The exhaust pipe was created after a picture found on internet for an Ariel atom car.

The pipe design made me use the sweep tool which I hadn't tried before.

There is also a" 3D sketch- tool" in Solidworks which is good and I thought that a similar tool could be expected from Creo, and that the exhaust pipe would create a good opportunity to try it.

Main tools used in the making of the engine:

Extrude, Sweep, Swept Blend, Round, Group, Plane

My own reflections
When using the Sweep- and Swept Blend-tool I got a bit confused, it was different from what I was used to. When sketching for a swept blend in Creo the profile and the path are in the same plane , which saves you time. The time that it takes to add a new plane, rotate the view and create a sketch in the new plane, which is good. Although it was not as intuitive as I expected it to be, in other programs you can easily visualize the result but in this case it's very hard to do so.

The "3D-sketch-tool" that I hoped to evaluate didn't exist. I searched after it for a long time under all the tabs and I goggled the web without finding it. I understand that Creo must offer some kind of comparable solution but finally I gave up. The result of this quest left me therefore quite disappointed.

6.6 Assembly – *Daniel Lissert*

At first I thought that when assembling the model a part that has been put into place cannot be moved around, it is locked in a fixed position. Because of this it was hard to understand how movement of one part would affect the rest of the assembly.

The program offers you the possibility to move parts in "assembly state" and there is a tool that can be used to move assembled parts, but when you are used to this feature you'll get irritated when assembling your model.

But then I realized that there was different mate-options that needed to be used to assemble parts with correct relations.

The conclusion is that it is hard to understand how to properly mate the assembly without instructions, but I believe that goes for all mayor CAD-programs.

6.7 Rear suspension - *Anton Lindberg*

General
The program is harder to use due to difficulties seeing lines and planes compared to Solidworks (SW). It also has some issues giving responses to the user, eg. when clicking somewhere it is hardly marked sometimes.

Functions are more "baked togehter" which makes the header look cleaner with less buttons. This is an okay way to navigate for basic functions but I believe it is harder to find the right functions for the task. An example is that Extrude combines both cut-extrude and extrude in a single function. Sweep combines different type of sweeps and functions in droplists.

If you are used to SW it takes a little time to get used to moving the model around with a regular mouse. I'd recommend using a 3D-mouse since scrolling with a regular mouse goes the other way compared to SW.

Model tree

The model tree is not as logic as in SW. It is harder to move order between features and change visibility, etc. Though it is possible to do basic functions through the tree.

Manikin

I haven't used the manikin plugin to help with constructing the rear suspension. Though, I've tried it in assemblies and it seems to be a really great function, something that SW is really missing. I think that it is much easier to estimate product sizing before making models with help from Manikin.

Change inches < - > millimeters

This function is easy accessible and easy to change 1 inch to 1mm or 1 inch to 2,54mm when making a part with the wrong dimensioning system. This function does not work really well within assemblies though because every part need to be changed.

Scale

Scale is good for parts not having any precise measures. This function needs to be made on every part within an assembly. Creo does not tell you that scale doesn't work so you have to measure and check dimensions after every try which is a little annoying. Some parts have problems scaling since they don't keep relations and gets distorted.

Sweep

Sweep is easy to use and possible to change distance between revolutions in a single sweep if making a helix sweep. There are some difficulties finding what's wrong when Creo decides not to finish a sweep. That is what I think is the most annoying problem with Creo; that you don't get as much information what is wrong as in SW.

Sketch

In sketch mode, it is quiet easy to get started. The functions look similar to SW and have about the same names. Though there are problems selecting edges on already existing parts when sketching and referring points to other sketches.

Fillets/rounds and mirror in sketch, etc. is easy to use and understand when familiar to SW.

Though I did not find any possibilities to make easy 3D-sketches as in SW. This makes it hard to

Sketch functions

Similar to SW except that it is harder to select edges on existing features and to relate sketches to already made parts. Also lacking of 3D-sketches.

Extrude

Easy to use and to understand. Though it also contains extruded cut which took a while to realize. All features in Creo can either add or reduce material. Something different from SW is that Creo sets its own dimensions to a feature when finishing it. This can make you forget to

set some dimensions on more complex parts since they already are showing a dimension. You can't see if the part is defined or not as in SW.

Assembly

The assembly mode is a little hard to understand at first. Mates are similar to SW but you need to set "mate mode" to for example "pin" to be able to move the parts. Compared to SW where parts are movable until perfectly mated. It is also hard to make new or change parts in assembly since there are problems sketching from already created parts.

6.8 Wheels - *Dina Thamir*

Tire: däck

Wheel: hela hjulet, fälg + däck

Rims: fälg

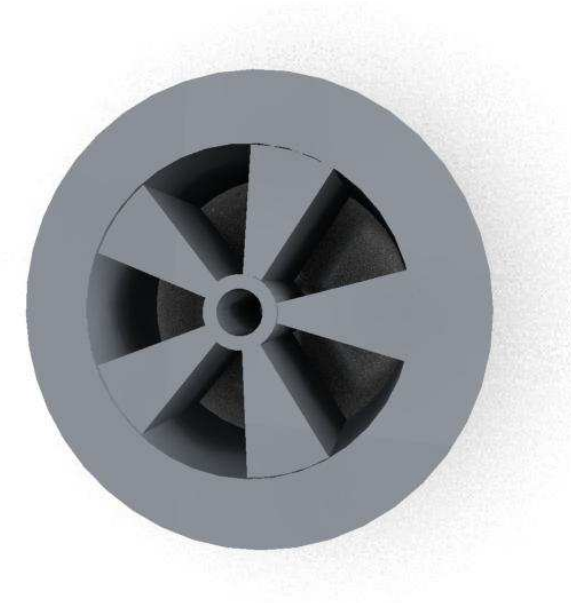
Studs: dubbar

Wheel pattern: spår

I have been assigned the crosskarts wheels. We determined the important dimensions of the car and then we would adjust our parts eventually. I planned the dimensions of the wheels together with Christian who worked with the front suspension and Markus who did the brakes. I constructed two different tires so that we could choose one of them later in the project.

I began by constructing a basic model to see which dimensions the wheels need to be, and then I made two more detailed models, a wheel with wheel-pattern and another with studs.

This is the first cad model, where it was determined that the diameter would be about 500 mm.



For this model I used the tolls Extrude and Pattern. Because it was basic I encountered no major problems.

After the measurements were determined, I began to construct the detailed model. I started with the construction of the rim, I drew the profile then I used the revolve tool with the center axis as a reference. The tire was constructed in the same way as the rim and in the same part.

For both wheel pattern and studs, I used almost the same rim and tires.

The studs were made first with a profile of one stud and then an extrude. After that I did a circular pattern for distributing the studs across the surface.

Wheel pattern were made using sweep and pattern to distribute them over the whole surface.

Finally the holes where constructed where the brakes would be fastened with five screws. The dimension of the hole is 18 mm. Also here pattern is used to distribute the screws evenly.

After a discussion with some course members we decided to choose the wheel with wheel-pattern because we thought that it fits more for the crosskart.

The final result:

Tools and reflection:

The tools I have used for the modeling of the wheels is:

Sketch: The sketch tool was similar to SolidWorks, the functions and the names where quit the same. though I had problem when I sketch on already existing features, it was hard to select edges for example

Revolve: It was quiet easy and similar to SolidWorks.

Extrude and cut-extrude: Easy and very similar to SolidWorks but it was hard to find out that extruded cut where ports on the same bottom as Extrude.

Round: It was similar to SolidWorks, but if you change the radius only the last round follows and not all. But if you select all of them at the same time then it will work easily.

Hole: at first I had problem understanding this tool, the hole where too small or too big for my model. Then I got help to change inches to millimeters and it worked, but you have to do it every time you open new part and it was hard to remember every time. Also it was hard to place the hole though you have to define where to place it in two directions.

Sweep: this tool was a little bit hard to understand at first, you had to go backwards in the process, press the sweep then go into the part and draw both profile and guide curve in the same sketch.

Pattern: Also this tool was easy to use but it took a very long time for all features to load, save, and open. also it took a while to realize that you have to "group" the extrude or the hole to use the pattern tool.

Render: it was very easy to use, the final render upload fast but as soon as you move the mouse the render disappers and you have to do it all over again.

In general, I thought the program was difficult to use comparing to SolidWorks. Mark, zoom and move was different from SolidWorks, it took some time before I could get used to using the regular mouse.

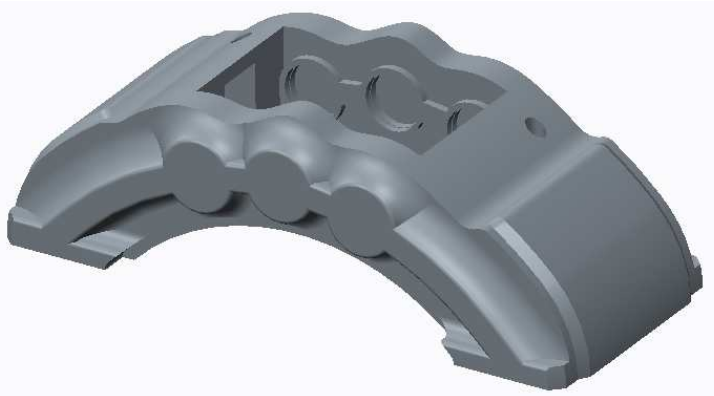
Some functions were difficult to find because they fall under the same tool buttom, such as the cut-extrude in the Extrude button.

Dimensioning I thought was most difficult because creo measures all edges automatically and you can only change a measurement, not remove it.

6.9 Brakes (Calipers) – *Markus Angell*

The brakes I have created are inspired by “Brembo Racing Brake Systems” and typical for Audi R8. The most important dimensions were included in their brochure, so I could create brake discs and calipers that would fit most cars. The dimensions I could not find in the drawings were improvised and by comparing the model with photos of the real components.

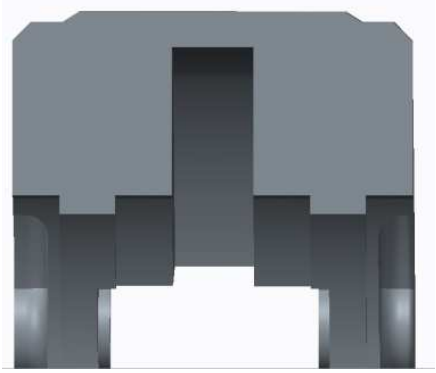
This is how the result looked like:



(Note that the brake pads are not added and the pistons not created as individual components.)

During the modelling of the calipers, which is the housing for brake pads and pistons, **the main tools I used were:** Sketch, Splines (Sketch), Revolve, Mirror, Group, Extrude and Round.

My own reflection on using Creo 2.0:



I started off by drawing a profile in the sketch mode and revolving it, which is shown in the picture to the right. The sketching tool is a bit problematic to handle at the beginning, compared to Solidworks which is very user friendly and understandable almost from the beginning. It took a while before I understood that the dimensions needed to be locked so they would not change when editing related dimensions. The upside with sketching in Creo is that it suggests lengths and relations which are already in the sketch and could shorten the time spent sketching, but the downside with this is those times where the relations are added (by default) when not desired. Splines seemed like a total free form sketch tool, where adjustment parameters were difficult or impossible to add, I chose not to spend much time with this way of sketching and used it for some sort of finish to add some shape to my calipers in combination with extruded cut (see the top of the model). The extrude tool was used in mainly three different areas, one of them already mentioned. Another area of use was the squared cut from top to bottom, and the last application to make the “rooms” for the pistons on the sides, which I later on rounded and

mirrored to get the same shape on both sides. In Creo features need to be grouped before being patterned or mirrored. The rounds I used on the sides were grouped before they could be mirrored, afterwrds they could be ungrouped if it is desire

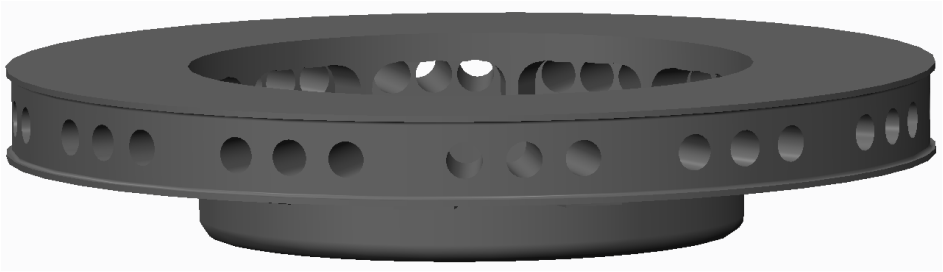
6.10 Brakes (Disc) – *Markus Angell*

The brake disc was created in the same manner as the calipers, using drawings as support in the modelling process. The disc is connected to an “axle-holder” which in its turn is supposed to be connected to the wheel.

The main tools I used were: Sketch, Pattern (Group), Extrude and Round.

My own reflection on using Creo 2.0:

The disc was created by extruding an outer circle downwards, and adding patterns from the circle’s midpoint outwards as seen in the picture below. The patterns are easily added by choosing the option “Pattern”, as in Solidworks it requires calculation to get the correct number of patterned features divided on the whole outer length.



The component under the brake disc was extruded as a dense feature and later on made hollow by adding an extruded cut. As a finish I added “Round” to the majority of the available edges. All in all I think that Creo 2.0 has a complex interface with more difficult usage of tools in comparison to Solidworks, but are manageable to learn over time. I have not used any tools which are not available in Solidworks, but I can still see the strength of being able to use software like Creo 2.0 as well, since a lot of businesses are currently using this for their CAD-operations.

6.11 Chassis- *Gabriel Anaya*

Introduction

¿What is a chassis?

The chassis is the vehicle´s skeleton. The main function of this structure is to provide strength, rigidity and shape. Besides, it holds the different mechanical parts of the car (engine, suspension, steering, etc.) making possible for them to connect each other. There are several types of car chassis. Usually, car companies choose the chassis type depending on the performances needed, the final price and and the kind of vehicle. Most of the premium segment sports cars have tubular chasis. This kind of structure provides high stability, and decrease the weight of the car. Nevertheless, due to its difficult design and manufacturing, the production cost of these chassis is very high. Therefore, tubular chassis´ cars are not mass produced.

Ariel Atom

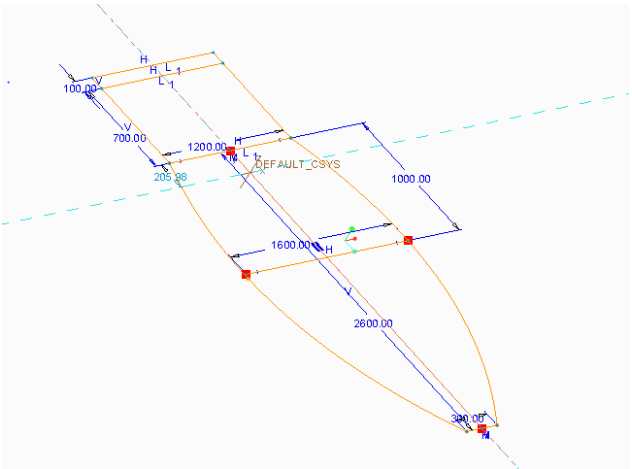
Ariel Atom´s mechanical performances are low, if we compare with other supersports cars. However, its performance on the track gets over the most part of them. This is due to two

important concepts: lightness and aerodynamic. Ariel Atom´s tubular chassis acts as a kind of exoskeleton, it doesn´t hold any bodywork on it. As a result, the car´s weight decrease a lot. Besides, the aerodynamic shape of the structure provides a low friction coefficient. These factors increase the Ariel Atom performance to the level of the top models from brands as Ferrari or Lamborghini.

Ariel Atom´s chassis shape is long and sharp-pointed. The structure has two cubes or cages, one in the front and the other in the back. They hold the suspensions, engine, transmission axis and other mechanical parts. These cubes are connected by 4 curved rods, which set the cabin and give the car its aerodynamic shape. In the middle part there is a inclined “wall” that separates the engine from the passengers. Upper and bottom parts are connected by truss rods. The anti-tip arc and central rods protect the passengers, because they keep the cabin shape in case of accident.

Design process

1.- Basic Sketch:



- We create a sketch (sketch 1) on the default plane called TOP. This sketch will be a template for the whole design process. It represents the contour of the chassis´ bottom viewed from above. Using the main measures of the real car (width and length), we draw a rectangle that represents the engine´s cage. Then we add two parallel lines in front of it and we connect the corners of the cage with the final point of each line using the Spline tool. The first parallel line (line 1) indicates the widest part or the bottom pipes, and the second one (line 2) sets the final tip of the car.

2.- First parallel plane:

-Using the Plane tool, we set a new plane (Plane1) parallel to the first one 300 mm far from it. This length settles the difference between the highest and the lowest point of the cabin bottom pipes. Then we open a new sketch (sketch 2) on this plane, and using the Reference tool we select the lines of the Sketch 1. After re-drawing the reference lines that have appeared, we draw two more splines (main splines), starting in the back corner of the engine rectangle and finishing in the tip. These splines are a template for the main pipes of the chassis.

3.-Inclined plane:

- It´s time to set the plane where the bottom pipes will be. To do it, we use two lines as reference. One of them is line 2 of Sketch 1, the other is the final line of the engine rectangle in Sketch 2. Then we set a new Sketch (Sketch 3) in this plane and we repeat the process in step 2. We select Reference to objects and copy the first splines. Now we have the bottom pipes´ reference lines on the right place.

At this point, it is important to notice how the Reference to objects tool works. It projects a copy of the selected object into the Sketch we are working on at that moment. Nevertheless, the direction of the projection is settled by the plane where the referenced object is.

Tools: sketch, plane, spline, reference to objects

4.-Front and back planes, reference plane on top:

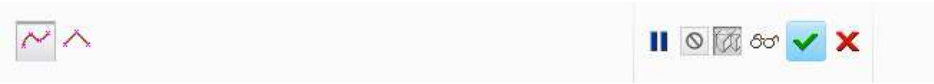
-We have just designed the references for the bottom of the car. With the next step we are going to create references for the front part, the back part and the top. For the front vertical plane we use the tip lines as references, and we divide the sketch in two levels. The lowest one will set the final part of the two top main pipes, the second is a reference for the highest plane. We repeat the operation in the back plane and then we create the reference plane on top. The distance between the lowest and the highest planes is 700 mm. We select the main splines as a reference for the highest plane´s Sketch. Now we have finished the “box” where the model will be placed in.

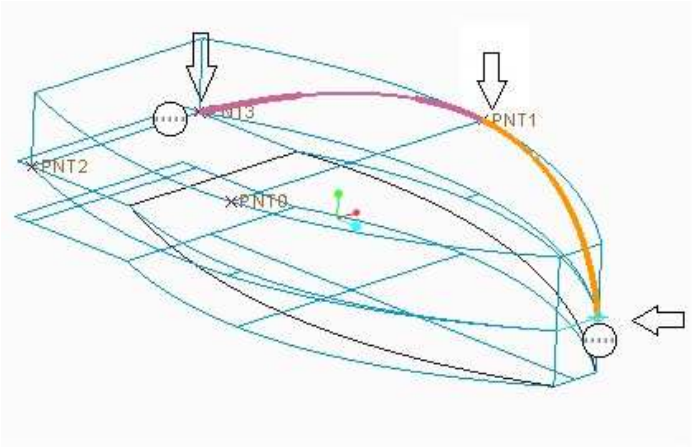
5.-Curved rod

-The two main curved rods are the most important bars of the car. They bring the aerodynamic shape and the rigidity needed. These rods start in the back part of the engine and cross the whole vehicle connecting the back, the middle and the front part.

The design of these rods is problematic. Although the curves are contained in a plane, is difficult to figure out how to set them. Fortunately, using the tool CURVE we can draw any 3D curve just selecting three points. This function doesn´t need any previous sketch.

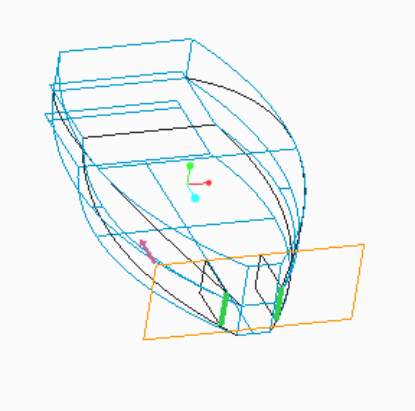
We select the three points, as shown in the picture.

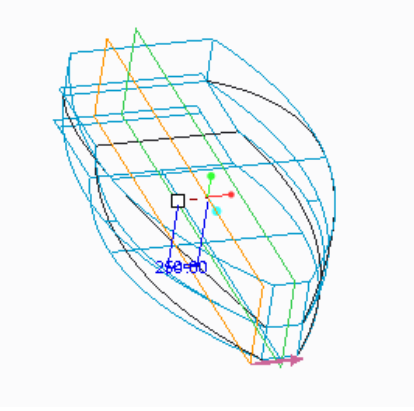




6.-Front cage

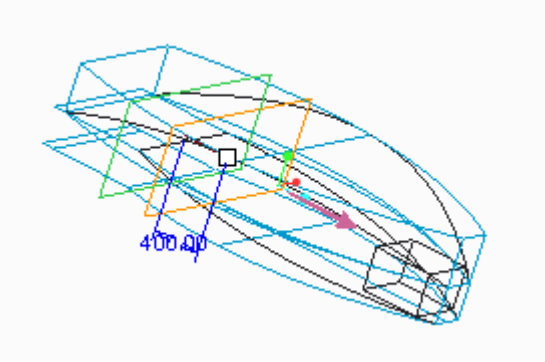
-We have already drawn the basic shape of the car, now it´s time to start designing most specific parts. We define a plane parallel to default Right plane, and we set two points in the place where the curved lines cut this plane. Then we open a new Sketch on that plane and draw a trapezoid figure starting from those points. We select OK and copy the figure to the other side of the car using the tool symmetry. Now we have all of the references needed to connect both figures by repeating the operation Plane-Sketch.





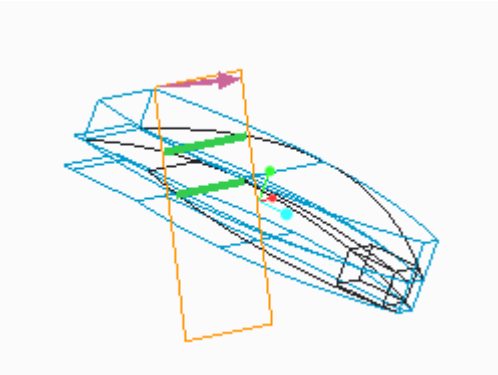
7.- Dividing wall

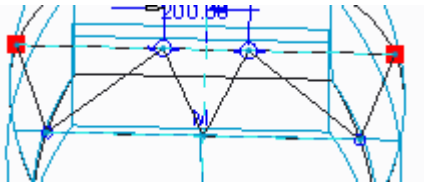
-This part separates the engine cage from the cabin. It is situated on an inclined plane. To set the reference lines needed, we create two parallel vertical planes in the middle of the car,



using the engine cage´s lines as reference for the position.

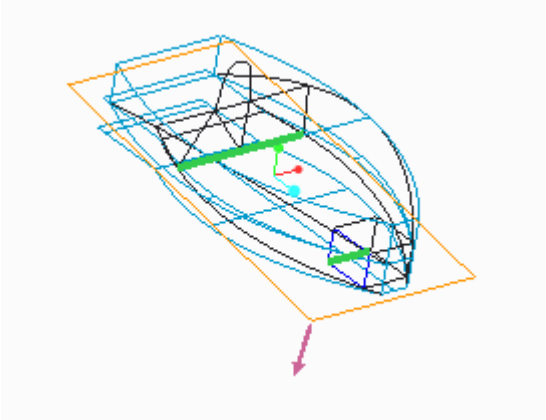
Then we put four points in the intersection between these planes and the curved lines. Now we connect the points by sketching two horizontal lines. These lines are the reference for the creation of the inclined plane. Finally, we draw the dividing wall on the inclined plane and the back roll bar on the vertical, as shown in the picture.

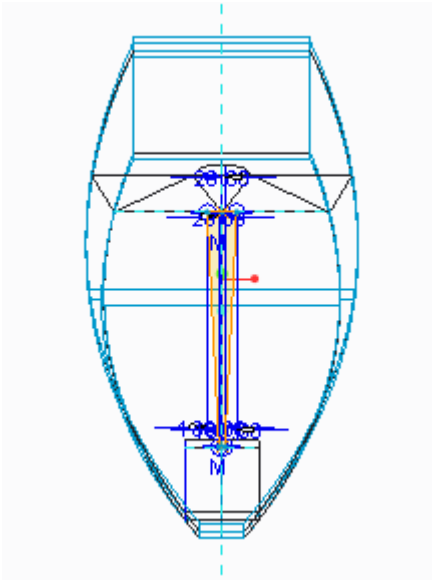




8.-Central rolls

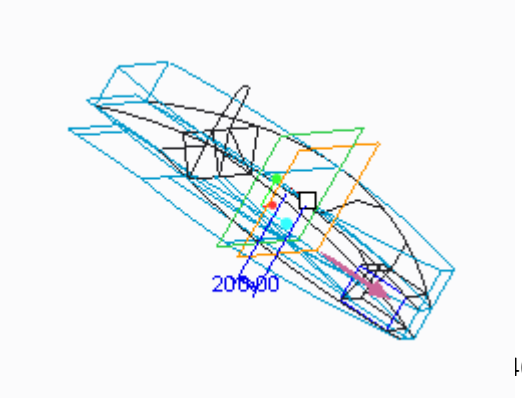
-These bars provide rigidity to the cabin. They are situated between the passengers´ seats in longitudinal direction. To set the plane we use the dividing wall´s lower bar and the front cage´s lower bar. Then we sketch the two lines on the required position.





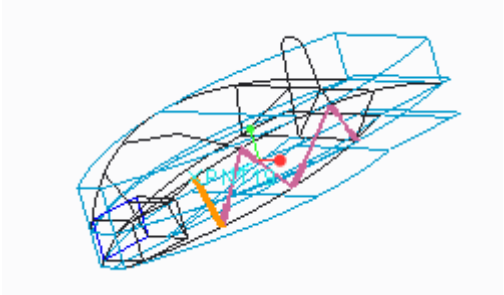
9.-Lateral cross- bars, suspension support bar, last connection rods

-The lateral cross-bars are intended to connect the upper and lower curved lines. To define them, first we have to set several points on these lines. This is done by adding vertical planes and intersecting them with

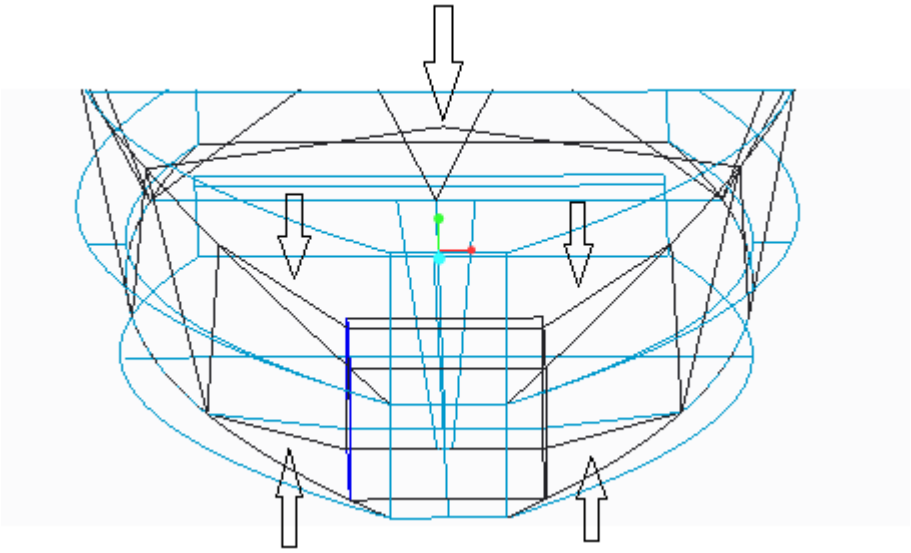


the lines. Now we select reference data=>curve => linked straight lines and connect the points. Finally, we select the lines and apply symmetry to copy them in the other side.

We sketch the suspension support bar on one of the vertical planes recently done (the central one). It is important to make the bars converge in the same node to avoid unnecessary stress. The support bar is pointed by the central arrow in the picture below.



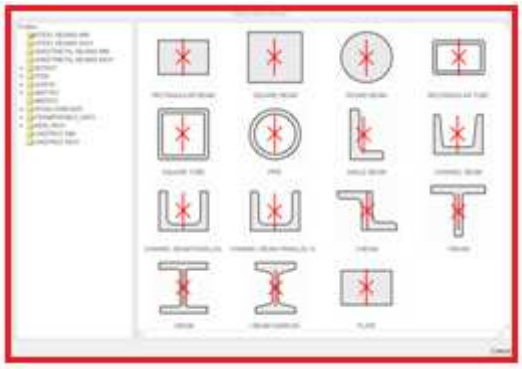
Finally, we add the last connection rods between the front cage and the lateral curved bars using the Curve tool (also pointed in the picture)



10.-Back cage

-The back cage contains the engine and supports the suspensions, so its measurements are very affected by the features of both parts. Therefore, the designer of these parts Anton Lindberg and Daniel Lissert modelled a cage that fit with them. Afterwards, I used the measures of this model to design the real chassis´ cage.

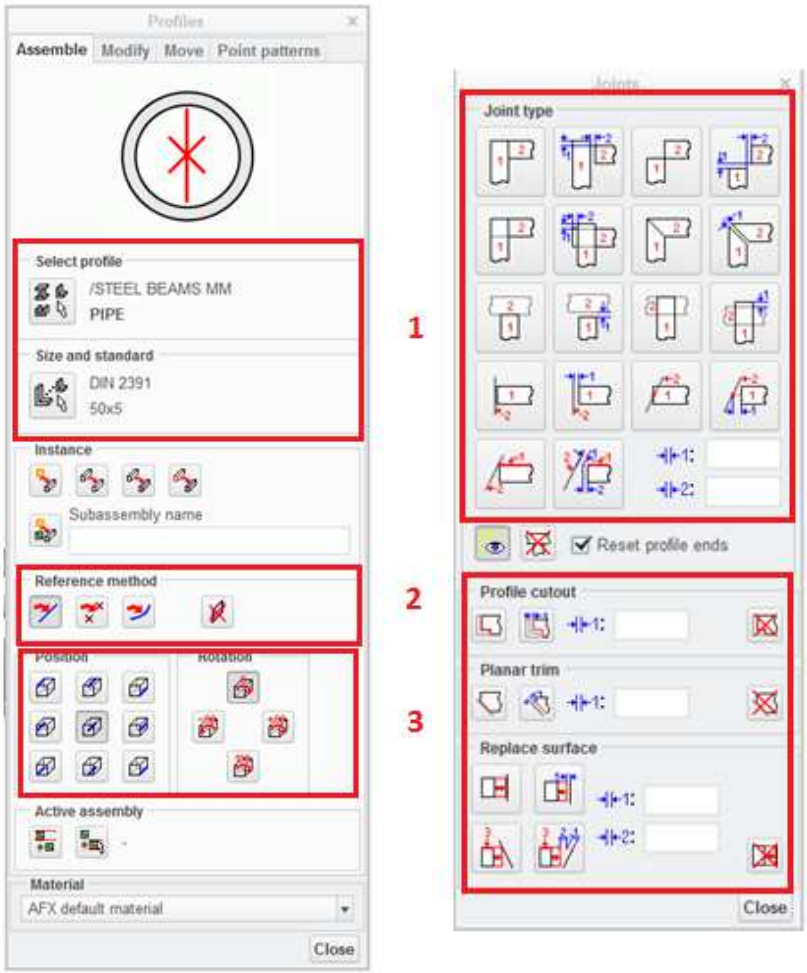
Advanced Framework Extension (AFX)



-AFX is a Creo´s extension which allow us to design complicated frameworks very fast. Its use concerns every single type of metal structure, such as building´s structures, high-tension towers, phone antennas or vehicles´ chassis.

In order to use this extension we have to work in an assembly file, because every added profile will be a new part for the program. To set a reference for the framework, we can either add a previous template to our assembly or either creating it directly in our file.

Once the template is settled, we select profile and it appears a new window with several options. This menu allows us to choose the correct profile by browsing through the tabs, and it is very intuitive. We can also select the standard, the size (1), the reference method (between points, following a baseline, etc.)(2), and the position of the profiles (3).

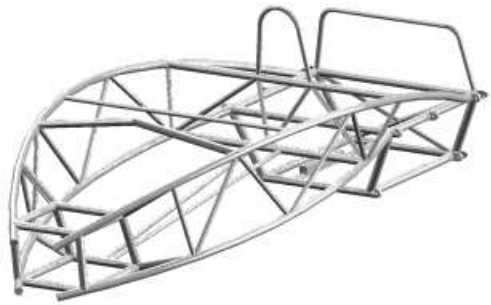


Now all we have to do is going back to the model and start creating the framework by selecting the point or the lines of our template. If we want to change the profile or any other feature, we can open the profile´s window again and do it very easily.

However, the program does not add the joints automatically.

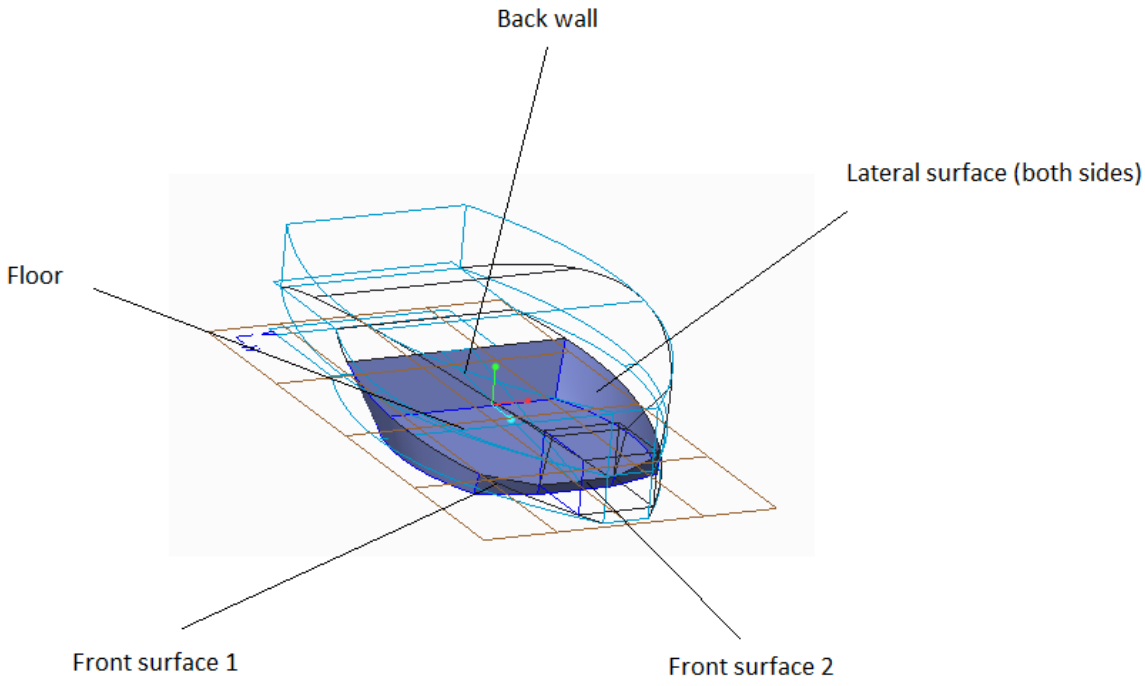
In order to creating the connections between profiles, we are going to use **Joints**. This tool is very time-saver, because it provides a way to execute complicated cuts, extensions and surfaces very fast. We select **Joints** and appears a new window with several tabs. Among them, there are 14 default connections (4) which we can add just selecting the tab and the profiles. Besides, there are another three options to execute all kinds of joints by selecting the most suitable **profile´s cut**, **plane´s cut and surface replacement (**5**)**. It is also possible to add bolts, connectors and pins.

Using all those features and tools, we create the structure and the joints. This is the chassis.



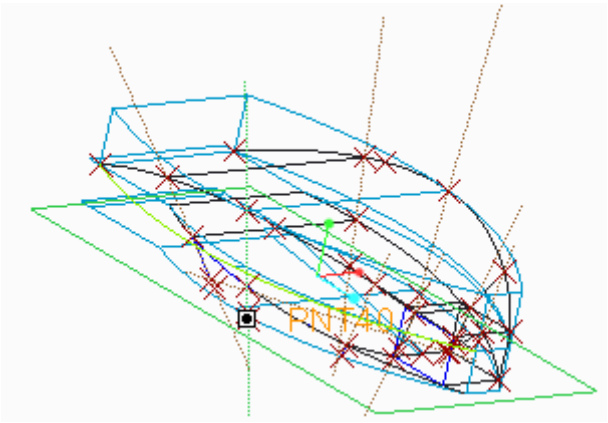
6.12 Bottom of the cabin- *Gabriel Anaya*

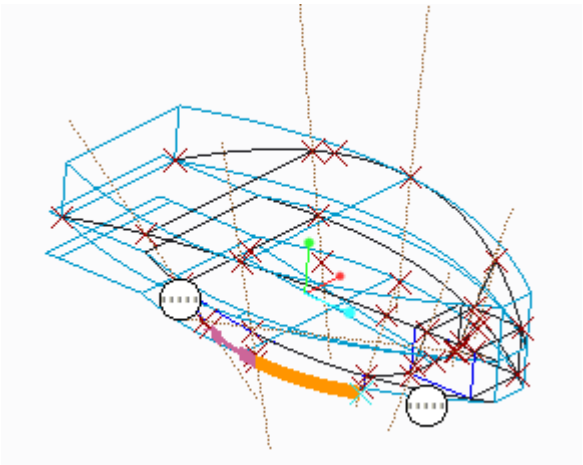
The bottom of the cabin is shell-shaped. It is composed of seven surfaces connected each other. The chassis doesn't have any bottom bar to hold the piece, so this part has to fit perfectly with the chassis in order to ensure a strong connection.



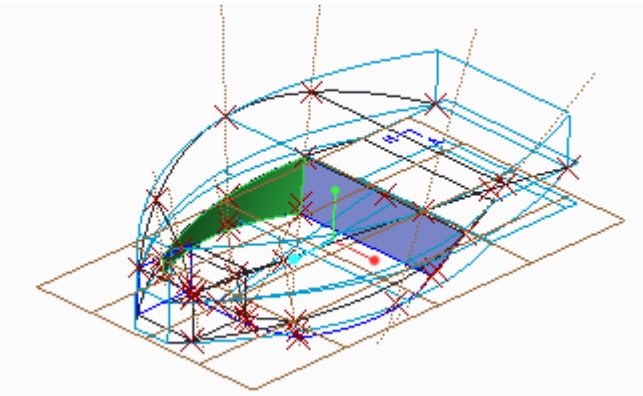
Design process

First, we create the floor's plane. The back wall is included in the dividing wall's plane. To set the references for the lateral and front lines, we draw axis connecting the upper and bottom chassis bars. The points where these axis pass through the base plane are the references. Now we connect the points using **Curves.**

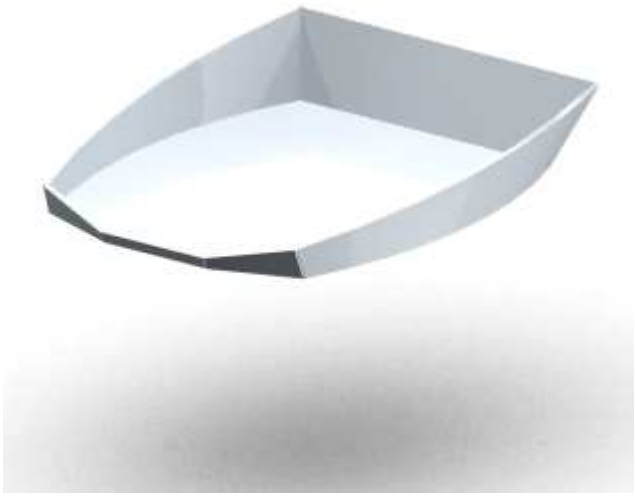




Now, we open **Style-> Surface** and select the lines around. We repeat the same process for all the surfaces. It's important to notice that the lateral surfaces are not flat, so we have to select it correctly in order to having a surface with the correct inclination.



Finally, we cut the upper part of the surfaces to make them fit with the chassis (tool **Extend**), and increase the thickness of the whole part using **Thickness.**



Personal opinion

Creo Parametric 2.0

-The interface of the program is not attractive, but the management of basic tools as display options, visualization options (pan, zoom, views, etc.) is easy. The file management is intuitive too. The model tree allows us to follow the design process, nevertheless, it´s necessary to select the **Edit Definition** tab of each step if we want to know all the measures used.

Sometimes there is a lack of references while sketching, but with selecting the **References** tab can we can reference to every surface, line, plane, point or edge.

However, when we look deeper in the program, the problems start. Tools as Extrude, Sweep, Pattern and Surface are easy to handle when we are working in a simple design. Nevertheless, when we want to reach a more advanced level, the menus become hard to manage. There is a good example for this: the assembly tool. When we just want to add a fixed part, the process is very easy, but if we want to add a mobile part it is necessary to set many features.

AFX

Advanced Framework Extension is a very useful tool. Display tab and menus to select profiles, joints or connectors are very intuitive. It is difficult to reach a expert level of management, but once we got it we can save much time, because this extension is able to execute complicated tasks automatically.

In conclusion, in my opinion Creo Parametric 2.0 is a very useful program for advanced users, but is not the most suitable one for introductory level.

7. Conclusions and recommendations

Windchill ofrece una interfaz algo compleja al principio, pero una vez aprendido el patrón de almacenamiento permite una rápida navegación entre los diferentes menús. Cada archivo almacenado en una carpeta tiene a la derecha información que nos permite saber si está siendo modificado por alguien, la versión en la que se encuentra, quien fue la última persona que lo modificó y cuando. Además, pinchando sobre el icono se nos muestra una ventana con una imagen del objeto, las piezas que contiene y en que otros productos se está utilizando. Aunque cortar y pegar archivos de una carpeta a otra es sencillo, copiarlos es bastante más difícil, y una vez definido su nombre inicialmente no es posible cambiarlo. La conexión entre Windchill y creo es muy buena, ya que una vez conectados al servidor en el árbol de modelo tenemos en todo momento dos pestanas que nos comunican directamente con las carpetas de Windchill. Además, no es necesario tener instalado el programa de diseño para seguir las evoluciones del proyecto, ya que se puede acceder al servidor de Windchill directamente por internet, desde cualquier ordenador. El sistema de guardado y acceso a los archivos es quizás la parte a la que más cuesta acostumbrarse. Las carpetas de Windchill son comunes a todos los usuarios, pero cada usuario tiene su propio espacio de trabajo independiente. Los archivos almacenados se encuentran por defecto checked-in, de forma que cualquiera puede abrirlos y trabajar en ellos. Sin embargo, al abrir un archive desde Windchill el estado del archive pasa a checked-out, y va a nuestro espacio de trabajo. En esta situacion, ningun otro usuario puede hacer cambios en el archivo. Tras finalizar la sesión es necesario cargarlo para que estos cambio se almacenen, y así poder seguir trabajando desde el mismo punto en la siguiente session. Durante ese tiempo el archive sigue bloqueado para los demás miembros hasta que la persona que lo está utilizando vuelva a almacenarlo.

La opinión acerca de la recomendación o no de Windchill está dividida entre los estudiantes de cursos avanzados y estudiantes de cursos intermedios del grupo. Sabiendo que, en algunos proyectos de la escuela es una sola persona la que realiza las labores de diseño por ordenador y los demás miembros se limitan a hacer observaciones y comentarios para mejorarlo, es estos casos el uso de Windchill podría ser redundante e innecesario. Sin embargo, los estudiantes que han estado involucrados en proyectos más avanzados, en los que el diseño se realiza entre varios miembros, opinan que para este tipo de trabajos el uso de un sistema PDM como Windchill puede ser muy ventajoso. Esto se debe especialmente a la facilidad que estos sistemas ofrecen para acceder a diseños y archivos de otros miembros, y asegurarse de esta manera del correcto avance del diseño y el encaje de las diferentes piezas sin mezclar archivos y evitando así el desorden y mezcla de archivos que se produce de otra manera. Actualmente en la escuela se utiliza un sistema de PDM de red interna, pero Windchill ofrece la ventaja de que, al estar en internet, permite realizar tareas desde cualquier ordenador que esté conectado a la red en cualquier lugar, mientas que en el PDM de SolidWorks es necesario estar en un ordenador de la escuela conectado a la red interna.

Por lo dicho anteriormente, recomendamos el uso y la adquisición de licencias de Windchill para proyectos de cursos avanzados, ya que en este tipo de cursos se maximizan las ventajas que éste tipo de programas puede ofrecer.

Windchill también funciona con SolidWorks. Según la web oficial de PTC, el módulo Windchill Product Manager for SolidWorks permite almacenar y manejar todo tipo de documentos SW. No obstante, no ofrece ningún programa de instalación ni ayuda al respecto.

MÄLARDALEN UNIVERSITY
SWEDEN

IDT INNOVATION DESIGN AND ENGINEERING

KPP 307-Project Course

Product and Process Development

Final Reflection

Gabriel Anaya

26 jun 2013

Examinator: Anders Wikström

In the beginning of the semester the workgroup consisted of nine IDT students and me. During the first meeting, Anders Wikström and Bengt Gustaffsson explained us the two main purposes of the course. The first goal was to evaluate the programs Creo 2.0 and Windchill 10.0 (both edited by PTC), in order to decide if they were suitable for academic purposes or not. Besides, we were supposed to check the differences between Creo 2.0 and SolidWorks (the most often-used design program in the IDT) and find out if the new software platform set any further improvement. The second goal was to write a personal reflection afterwards, explaining and analyzing the development of the course by using the Gibb´s reflection model.

After several meetings, we decided the best evaluation method. These programs were supposed to be used in projects. Thus, the best way to do a useful analysis was to carry out a real project in order to reproduce the steps that other groups could take, and find out the potential problems the programs could have.

During the first month we follow several tutorials and took some lessons about both software tools. This was in order to get familiar with the programs management and basic features. Besides, we met once per week to discuss project issues: the path we would like to follow, the phases, dates, etc. Due to my poor knowledge about SolidWorks, I took tutorials about this program too, separately from my colleagues. By the end of the month, Bengt offered me the post of Project Manager, because of my deep implication in the project. I accepted the task.

Once we finished the introductory part of the project, we took a step further and started to design an engine. Bengt gave us all the necessary documentation about a radial engine he had designed before with SolidWorks. During the second month, each group member modelled his own engine using Creo and Winchill. During this period, I kept close contact with my colleagues in order to follow the evolution of the project, help anyone who would have any problem and ask for help when I needed it. By the end of this month, I suggested to carry out an evaluation to reflect our first impressions. Therefore, we designed a table to mark the features of the program (from 1 to 5) and filled it.

During the first and the second parts we reached the necessary level to develop the main project phase. At this point, there was a different involvement level between the group members. This was due mainly to the different number of extra courses we all had. Therefore, we discussed about the possibility of dividing the group in several “sub-groups”, in order to give everyone an amount of work that he or she could handle to contribute. Nevertheless, finally we decided to keep working altogether in the same product. We chose to model the Ariel Atom, an exoskeleton based high-performance car. We divided the car in 11 parts, giving one (or two) of them to each group member. Obviously, some parts were more difficult than others, so we delegated the tasks depending on the available time each person had to focus on the project. I chose to chassis, the main frame of the car. This part is designed to give the car strength and structure, and to hold all the other parts.

To develop this final project, information sharing and constant communication were mandatory in order to make all the parts fit. Therefore, we created a new group schedule to set hours to work altogether in the same computer room. Nevertheless, the attendance to the

meetings was irregular, due to extra workload. It was very important for me to follow the evolution of my colleagues´ work, not just because of my charge, but also because of the nature of my task. Each time I started modeling a new chassis´ part I had to ensure that it fit with those modeled by another group member. Fortunately, Windchill discovered itself as a really useful tool for this task. Once we started to see the project final approaching, we met to discuss about the evaluation report. We decided to divide the task: some group members would explain the project introduction and general description and the rest would write the final evaluation about both programs. Besides, each group member would describe shortly his own work and personal opinion about the tools he would have become more familiar with.

It was hard for me to find a project related with my field of study. The one I finally could join was not in the catalogue for exchange students. I was the only foreign member in the group, I did not speak Swedish and I did not have much experience working with the software program the other members were familiar with. This situation made me feel nervous in the beginning. I hesitated about my potential contribution to the project under these circumstances. Would I be able to adapt myself to the work method fast enough? Could I establish a proper communication with my colleagues?

All this thoughts made me increase my commitment with the course. As a matter of fact, I guess Bengt noticed it, because he offered me the post of Project Manager two weeks after the beginning of the course. This post made me feel more responsible about the final result of the project, but the fact that one of the responsible of the course gave me that task removed all the doubts and anxiety that I had in the beginning. Nevertheless, I was a bit worried about the reaction of my colleagues. Although I was the less extra-loaded member in the group, obviously I was also the less-experienced one and the newcomer. Soon I realize these thoughts were nonsense, because all my colleagues understood my situation and supported me. Besides, in that moment the course was a bit stuck and I realized that my new situation could give me the possibility to push the project forward.

During the following months I felt a bit disappointed. The general commitment level in the group was not as high as I would have liked. Most of the times, it was just some of us who took the important decisions; the assistance to meetings and to work sessions was irregular. Afterwards, I realized that many group members had several extra courses, so they could not focus on the project as much as I did. Nevertheless, it was very impressive for me to realize the great amount of job we could do when we worked altogether as a team. In that situation, every one contributed with their own opinion and skills, the ideas flown and we found solutions to the problems much faster than working separately. Finally, all the people did their job, and I felt satisfied at both individual and group level.

This has been a very enriching experience for me in many ways. Professionally, I have increased my knowledge about Product and Process Development in areas that I had never studied before. Besides, I have acquired good user level in two engineering programs used by companies worldwide.

Academically speaking, I had the opportunity to know a new learning methodology. The Spanish methodology is focused on the theoretical knowledge. The students are supposed to gain the practical knowledge once they are working in their posts. As a matter of fact, many of

them suffer a “job-shock” when they start their professional activity, and they need a hard period to adapt themselves to the task requirements. By contrast, in the Swedish academic system the students are supposed to be responsible about their own activity since the beginning. The teacher-student relationship is almost horizontal, so they have more freedom but they have to find their own resources to solve the problems, and take their own decisions. I guess this system makes smaller the gap between academic and professional environment.

On a personal level, by working in a new environment, I have become a more resourceful person. Especially during the initial and the final parts of the course, I had to do my best to overcome several kinds of challenges. The most important obstacle I found was about the communication with my colleagues. I was the only non-Swedish speaker in the group. Even though we all had enough English level to communicate, during the meetings the conversation often turned back to the Swedish by inertia, due to the difficulty to talk about a technical issue by using a language you are not fluent with. However, some colleagues made ongoing efforts to let me know about the main issue of the conversations, and to turn them back to the English. Thus, I always had the opportunity to contribute by giving my own point of view. Besides, I kept a parallel and close contact with some group members during the whole course in order to know everyone´s impressions and thoughts. As a support, we also created a group on Facebook where we uploaded the main information constantly (schedules, meetings, important data and links). Thanks to all these factors, the information exchange was very good.

I have gained self-confidence with this experience. After becoming Project Manager, I had several handicaps and challenges to overcome in order to make a good contribution to the workgroup. Therefore, since the beginning I tried to be honest and open-minded with my colleagues and let them know about my situation. I noticed that this kind of leadership is probably the best one to create a good work environment, where everybody offers support and ask for help when needed. Thus, the most important decisions were always taken between several members, reaching common points.

In conclusion, to list the most positive points, I did my best as a Project Manager, and I feel particularly satisfied because of the initial challenges I overcame. Anyway, I guess the final evaluation about my management should be done by my colleagues. About my work as a project member, I worked hard and never hesitated to take difficult tasks. I adapted myself to a new work environment and I would say that generally we all applied successfully the necessary methods to solve the problems.

Nevertheless, if I were in the same situation again, I would try to get previous information about the group members schedule availability, in order to set a better planning. I would also try to focus on the evaluation rather than on the product.

CREO evaluation

	Dina	Bashar	Daniel	Christian	Nur	Fredik	Anton	Marcus	Gabriel	Mean
Basic tools										
Create new file										
select template			5	3		2			3	3,3
selet directory		3	3	3					4	3,3
save	3	5	3	5	4	5			5	4,3
open	5	5	5	5	5	5			4	4,9
Interface design						2			1	1,5
Manage visual tools										
zoom	3	2	4	4	4				4	3,5
rotate	4	3	3	5	3				4	3,7
pan	2	4	4	4	3				4	3,5
select view	3	3	2	3	3				3	2,8
Display toolbar										
planes	4	5	3	5	4	5			4	4,3
axis	4	5	3	5	3	5			3	4,0
points	4	5	5	5	4	5			4	4,6
Csys		5		3		5				4,3
Sketch										
new sketch	4	3	3	4	3	5			5	3,9
add profile										
Normal geometri	3	3	3	3	3	3				3,0
Pallete			2	4		4			4	3,5
dimensions	1	1	3	3	2				3	2,2
measurement	2	2	3	3	2	2			2	2,3
Referense	2	3	2	4	3				4	3,0
Note									4	

Extrude/Cut											
	Intuitive use		3	4	4		4			4	3,8
	Revolved	4		3	3					3	3,3
	Sweep	3	1	3	5	2					2,8
	Loft		1	2							1,5
	Holes	2	5	1	5	4	4			4	3,6
	Rounds	5	5	3	5		2			5	4,2
	Chamfer	5	5	3	5	4				5	4,5
	Group	5		4			4			4	4,3
	Mirror	4		1	4	3	5			5	3,7
	Pattern										
		axis	4	2	5	3	3				5	3,7
		direction	4	4		3					4	3,8
Note										4	

Modeltree											
	Follow creationprocess	3	3	3	4	3	4			2	3,1
	change parameters	3	2	3	4	3	3			2	2,9
	Supress	3	3	3	3	3	2			4	3,0
Note										3	3,0
Insert referense											
	Plane			4	4					4	4,0
	axis			4	4					3	3,7
	point		3	3	4					4	3,5
	Coordinate system										
Note										4	
											3,5